Digital Photogrammetric System

PHOTOMOD

Version 6.0.2

USER MANUAL

DTM Generation

RACURS

Table of Contents

1. Purpose of the document 5
2. DTM Generation 5
3. Regular grid of nodes 5
    3.1. Grid creation 5
    3.2. Grid parameters 7
4. Pickets 9
    4.1. Pickets creation 9
        4.1.1. Pickets creation modes and methods 9
        4.1.2. The 'Pathway mode' toolbar 10
        4.1.3. Pickets creation in pathway mode 10
        4.1.4. Parameters of pathway mode 12
    4.2. Pickets automatic calculating 14
        4.2.1. Automatic calculating of pickets 14
        4.2.2. Correlator configurations 20
        4.2.3. Setup of correlator configuration 22
        4.2.4. Additional correlator's parameters 25
        4.2.5. Parameters of first approximation calculation 27
        4.2.6. Pickets calculation in distributed processing mode 28
    4.3. Pickets editing 30
    4.4. Filtering 31
        4.4.1. Pickets thinning 31
        4.4.2. Pickets filtering by Z-range 32
        4.4.3. Median pickets filtering by Z 33
        4.4.4. Filter of adjacent point objects 33
        4.4.5. Buildings and vegetation filter 34
        4.4.6. Filtering of linear objects 41
        4.4.7. Surface objects filter 42
5. Triangulation irregular network (TIN) 43
    5.1. The 'TIN' menu 43
    5.2. General information 44
    5.3. TIN creation 45
        5.3.1. Creation pre-regions for a block 45
        5.3.2. Loading base layers for TIN creation 48
        5.3.3. TIN borders creation 52
        5.3.4. Building TIN 53
    5.4. TIN display 54
    5.5. TIN loading 56
    5.6. TIN saving 57
    5.7. TIN info 57
        5.7.1. Brief TIN info 57
        5.7.2. General TIN info 58
        5.7.3. TIN area info 59
    5.8. TIN re-building 60
    5.9. Insert objects into TIN 62
    5.10. Accuracy control of TIN creation 62
        5.10.1. Check against adjustment points 62
        5.10.2. Check TIN against vector objects 64
        5.10.3. Verifying of TIN topology 65
    5.11. Export TIN 67
        5.11.1. Export TIN to DXF 67
        5.11.2. Export TIN to CSV 68
    5.12. Filtering 70
        5.12.1. Simplifying 70

5.12.2. Peak filter 71
5.12.3. TIN filtering by Z-range 72
5.12.4. Smooth TIN interpolation 73
6. Contour lines 75
6.1. Generation contour lines by TIN 75
6.2. Contours creation in batch mode 76
6.3. Generation contour lines by DEM 78
6.4. Contours creation from smooth DTM 79
6.5. Contour lines editing 82
6.6. Operations with contour lines 82
6.6.1. Saving contour lines 82
6.6.2. Contours merging 83
6.6.3. Check of contours merging quality 85
6.7. Accuracy control of contours creation 86
6.7.1. Control of contours crossing 86
6.7.2. Check contours by points 87
7. Digital Elevation Model 89
7.1. The ’DEM’ menu 89
7.2. General information 90
7.3. DEM creation 91
7.3.1. DEM creation by TIN 91
7.3.2. DEM creation using regular pickets 93
7.3.3. DEM creation using pickets 94
7.3.4. DEM creation using CSV-file 96
7.3.5. DEM creation using smooth model 97
7.3.6. Creation of dense DTM 99
7.3.7. Creation of dense DSM 105
7.3.8. DEM creation in batch mode 107
7.4. DEM filtering 110
7.4.1. Buildings and vegetation filter 110
7.4.2. Median filter 115
7.4.3. Smooth filter 116
7.5. Transformation of DEM coordinates 116
7.5.1. DEM transpose 116
7.5.2. Transformation of DEM coordinate system 117
7.6. Null cells in DEM 120
7.6.1. General information 120
7.6.2. Filling null cells using linear interpolation 120
7.6.3. Filling null cells using smooth interpolation 122
7.6.4. Filling null cells by constant value 123
7.6.5. Converting cells to null 124
7.7. DEM editing 125
7.7.1. DEM merging 125
7.7.2. Change DEM parts elevation 128
7.7.3. DEM shift 129
7.7.4. Split DEM into sheets 129
7.7.5. DEM interpolating 131
7.7.6. DEM cutting by polygons 132
7.7.7. The ’DEM editing’ toolbar 134
7.8. Accuracy control of DEM creation 135
7.8.1. Creation of difference DEM 135
7.8.2. DEMs comparing 136
7.8.3. Search for null cells 138
7.8.4. DEM control by TIN 139

7.8.5. DEM control by vector objects 141
7.8.6. DEM control by triangulation points 142
7.9. DEM saving 144
7.10. DEM recovering 146
7.11. DEM loading 146
7.12. DEM export 148
7.12.1. Convert DEM to pickets 148
7.12.2. DEM export 149
7.12.3. Export to CSV 149
7.12.4. Export to DTED 150
7.12.5. Export to ERDAS Imagine 151
7.12.6. Export to GeoTIFF 152
7.12.7. Export to PCIDSK 153
7.13. DEM import 155

1. Purpose of the document

This document contains detailed information about generation of digital elevation model (DEM). The document describes creating of regular nodes grid and working with layers needed for DEM generation pickets layers, triangulated irregular network (TIN), contour lines and DEMs. The document also describes main ways of generation of digital terrain model using these layers.

2. DTM Generation

The system provides possibility of generation of digital terrain model using different data sets.

Digital Terrain Model (DTM) – digital cartographic presentation of terrain surface both as regular grid of elevations (DEM) and as triangulated irregular network (TIN).

The following data sets (individually or in combination) are used for generation of digital terrain model:

- *Pickets* – point vector objects, located on relief surface;
- *Triangulation Irregular Network* (*TIN*) – one of the models of spatially coordinated data, used during designing of digital terrain model as elevation points in nodes of irregular network of triangles, that corresponds to Delaunay triangulation;
- *Contour lines* – vector lines, that connect points with the same terrain altitude;
- *Digital elevation model* – digital cartographic presentation of terrain surface as regular grid of elevation values.

It is possible to acquire data necessary for DTM generation both from central projection images and from satellite scanner images.

3. Regular grid of nodes

3.1. Grid creation

The system provides possibility of regular *grid* creation with specified step. The grid is used for pickets creation in automatic mode with help of correlator or in semi-automatic pathway mode. A grid step is used as a frequency of nodes creation, in which vicinity spatial coordinates are calculated and pickets are created. Moreover, the grid is used for different operations, such as map creation in stereo quality, DEM generation using dense model and for other operations (see the chapters Automatic pickets creation, Pickets creation in pathway mode in the 'Vectorization' User Manual).

The grid could be created both for the whole images block and for any part of a block or for selected stereopair. A shape of grid borders could be rectangular or as arbitrary polygon.

To create a grid select **Grid › Create** (**Ctrl+N, G**) or click the button of the main toolbar. After that a new layer *Grid* is created in the *Layer Manager*.

Area of grid creation is determined in one of the following ways:

- in order to create a grid with specified *rectangular border*, press and hold the **Shift** key, and drag by mouse a rectangular area over images block or over selected stereopair;
- in order to create a grid with *arbitrary border*, define a grid creation area in group selection mode by polygon and while holding pressed the **Shift** key, specify by mouse all nodes of boundary of a grid creation area. Complete creation of arbitrary grid border by mouse double click.

  It is possible to use pre-regions for grid area detection.

- to use vector layer polygons as areas of grid creation, perform the following actions:

  1. Select **Vectors › Create layer** to create vector layer without classifier or load layer with polygons which could be used as a grid boundaries.
  2. Create a polygon in such a way that its boundary coincides with boundaries of area which is used for grid creation.
  3. [optional] Select polygons to be used as boundaries for a grid creation, otherwise, a grid will be created considering all polygons of the layer.
  4. Select **Grid › Create boundaries from vectors**. After that created grid boundary coincides with outline of created polygon.

- to create a grid for the whole images block without defining its boundaries, select **Grid › Properties**, specify its parameters and click OK. Grid is created automatically for the entire block and the grid boundary passes over external boundary of block images.

When a grid boundary is defined the system creates a grid of nodes with specified or default parameters.

To save created grid in active profile resources use **Grid › Save** and **Grid › Save as** menu items. The grid is saved to the `*.x-grid` file.

To open existing grid, select **Grid › Open**. To close a grid, select **Grid › Close**.

Right-click menu of the *Grid* layer in the *Manager* duplicates save and load commands in the **Grid** menu, and allows to close a grid layer.

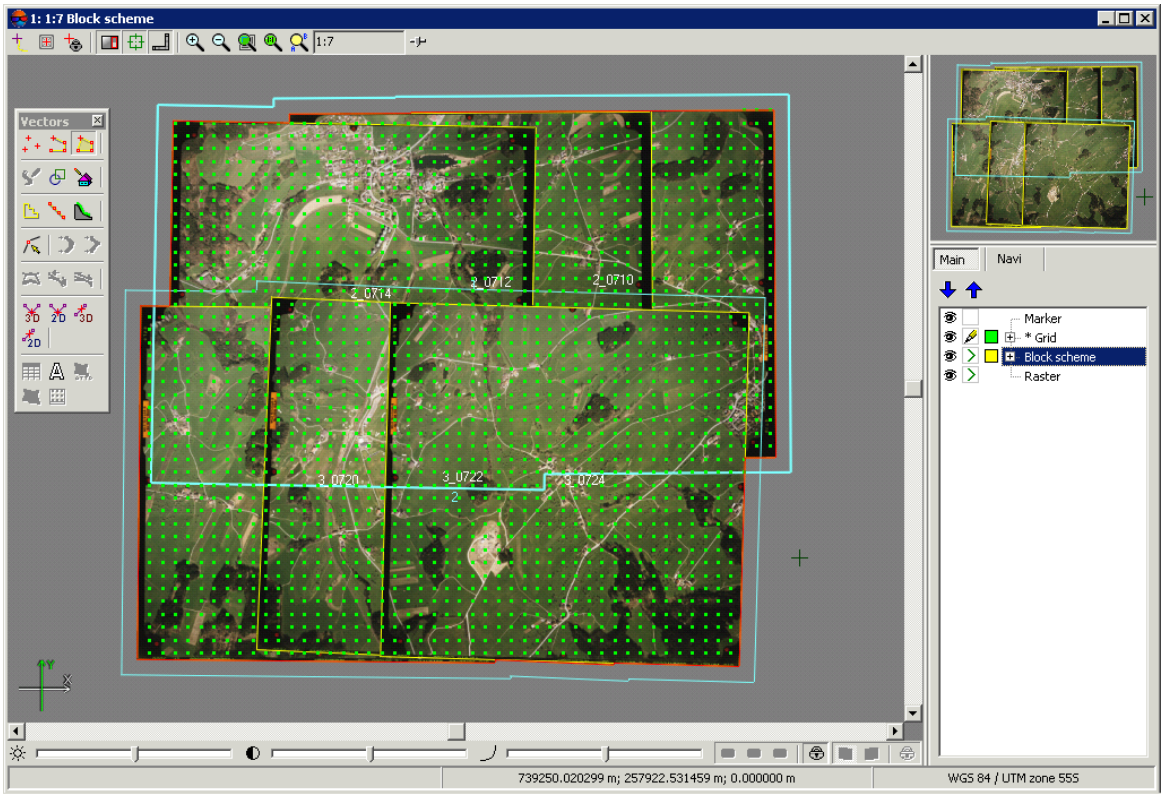


Fig. 1. Rectangular grid for the whole images block

3.2. Grid parameters

In order to change parameters of created grid, select **Grid › Properties**. The **Grid properties** window opens.

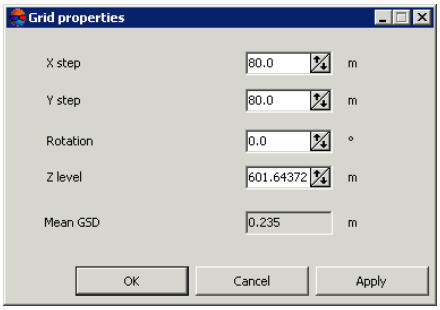


Fig. 2. Regular grid parameters

The **Grid properties** window allows to setup the following parameters:

- **X step** and **Y step** – steps for nodes creation correspondingly by X and Y axes in meters;

To change a direction of grid nodes tracking in the pathway mode specify negative values of grid step.

- **Rotation** – angle of nodes grid rotation in degrees;

Should be specified for manual work in the pathway mode.

At that, the nodes **grid only** is rotated. Area of nodes grid creation remains in initial position.

- **Z level** – a grid elevation above relief level (in meters) for visual correct display of the grid;
- **Ground sample distance (GSD)** – shows a value of average pixel size in meters, if a project contains adjustment results or pixel size is specified for at least one image.

To display a grid with specified parameters click the **Apply** button. To change parameters click OK.

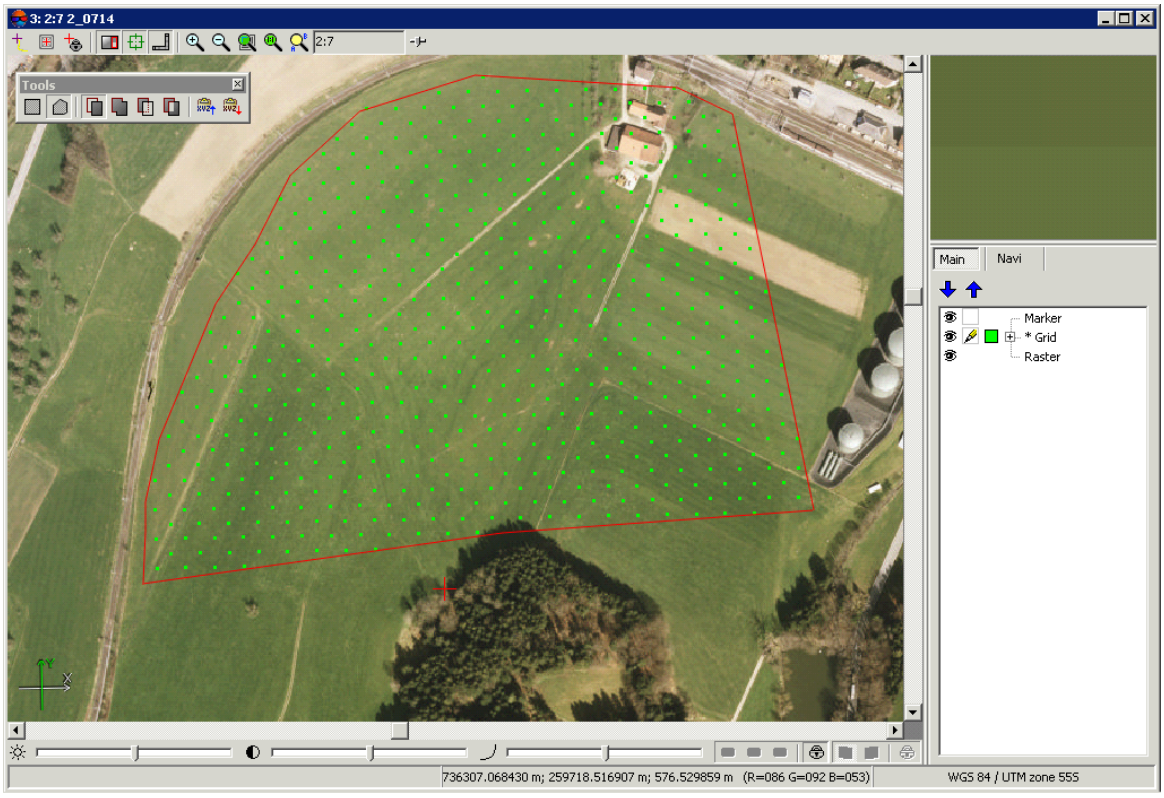


Fig. 3. Grid created using arbitrary border (with grid rotation)

In order to change grid display parameters click colourful square to the right from the layer in the *Manager*. The **Layer display settings** window opens. It allows to setup

color of grid nodes and borders, as well as a type and size of symbol used to show grid nodes.

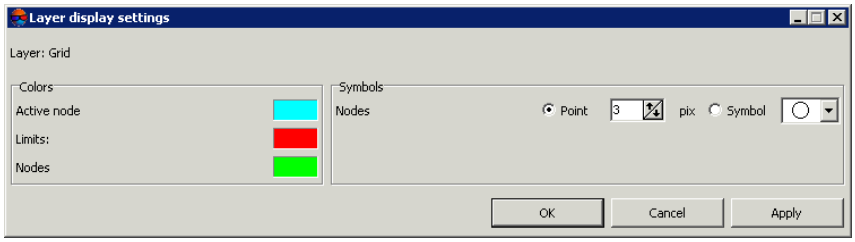


Fig. 4. Parameters of grid display

Additional parameters of grid display, as well as settings of pathway mode are setup on the **Grid** tab of the **Settings** window (see the 'General system's parameters' User Manual).

4. Pickets

4.1. Pickets creation

4.1.1. Pickets creation modes and methods

Pickets – point vector objects, located on relief surface. Pickets like other vector layers are used as a base layer for TIN generation.

The system allows the following modes of pickets creation:

- *manually in stereomode* – points are added manually in stereomode using correlator or without it;

  Correlator's parameters are setup in the window of general system's settings on the **Correlator** tab (see the 'General system's parameters' User Manual).

- *in semi-automatic pathway mode* with preliminary creation of regular nodes grid – that means sequenced passing of all nodes of regular grid and adding of pickets manually using correlator or without it, or skipping a node and passing to the next one;

  Correlator's parameters are setup in the window of general system's settings on the **Correlator** tab (see the 'General system's parameters' User Manual).

- in automatic mode with preliminary creation of regular nodes grid. At that in grid nodes vicinity the system calculates points spatial coordinates with help of correlator and creates pickets if the coordinates are calculated successfully. In automatic mode the

system passes grid's nodes specified number of times with different correlator parameters, performs accuracy control, pickets filtering, and saves quality evaluation.

For automatic pickets calculation correlator's parameters should be adjusted in the **Compute points** window.

4.1.2. The 'Pathway mode' toolbar

For work in pathway mode the system provides the **Vectors › Pathway mode** menu items, that are partially duplicated by the **Pathway mode** additional toolbar buttons.

Table 1. The 'Pathway mode' toolbar

Buttons	Function
Activate	allows to turn on/off pathway mode
Add point (**Enter**)	allows to add picket to current node
Bypass point (**Delete**)	allows to bypass a node (without adding picket) and to pass to the next node
Return (**Backspace**)	allows to return to the previous node (without picket deletion)
Marker to current grid node	allows to move marker to current active grid node
To nearest grid node	allows to move marker to nearest grid node of markers current position. The node becomes active automatically (provided that marker is inside grid in geodetic coordinate system), that allows to start passing pickets manually
To first grid node	allows to move marker to left lower grid node and to make the node active
Mode: scan rows	allows to turn on sequential mode of grid passing by rows
Mode: scan columns	allows to turn on sequential mode of grid passing by columns
Mode: 'zigzag' by rows	allows to turn on 'zigzag' mode of grid passing by rows
Mode: 'zigzag' by columns	allows to turn on 'zigzag' mode of grid passing by columns
Settings	opens the system settings window on the **Pathway mode** tab to setup the mode parameters (see Section 4.1.4)

4.1.3. Pickets creation in pathway mode

For *semi-automatic pickets creation* using regular grid the system provides **Pathway mode**.

In the pathway mode the system passes grid nodes in such a way that XY coordinates of points are measured uniformly. Marker XY coordinates are preserved while passing

from one grid node to another, and elevation is measured manually. Pickets measurement in pathway mode is possible both using correlator or without it.

The first coordinate of marker elevation during passing to the first node corresponds to grid level by Z, specified in the **Grid properties**. Marker's elevation level during passing to each next node is defined by settings.

Pickets adding in pathway mode is performed in stereomode in stereopair 2D-window.

If a grid is created for the entire images block, the system allows to setup automatic pass to the next stereopair while passing nodes. To do this use the window of general system's parameters on the **Pathway** tab.

To create pickets in pathway mode, perform the following actions:

1. Create a grid for required area of pickets creating.
2. Create or make active a vector layer for pickets creation.
3. Choose **Vectors › Pathway › Activate** or click the button of the **Pathway mode** additional toolbar. Marker will be placed to left lower grid corner – start point of passing.

   If grid node was not created before, it is created automatically for the whole images block.

   When the grid layer is closed the **pathway mode** is turned off automatically.

4. Select a direction of grid nodes passing using **Vectors › Pathway** menu or using buttons of the **Pathway mode** additional toolbar.
5. Choose **Window › New 2D-window (stereopair)** (**Ctrl+Alt+L**) or click the button of the main toolbar. The stereopair 2D-window is opened and allows to start grid nodes passing.
6. [optional] For image automatic centering by marker during passing to the next grid node turn on the **Fixed marker mode**. To do this click the button or use the **F6** hotkey.

   To vectorize objects during grid nodes passing in pathway mode, turn the **Fixed marker mode** temporarily off.

7. Click the to turn on stereo mode in 2D-window of a stereopair.

   To work in pathway mode it is necessary to turn on the mode of point objects creation – using the button of the **Vectors** additional toolbar.

8. To perform grid nodes passing the system provides the following tools:

   - to add picket to current grid node manually, set marker elevation using mouse wheel and press **Enter**;
   - to add picket to current grid node using correlator, press **Space** key to set marker elevation using correlator and press **Enter**;

     After the picket is added marker moves to the next grid node according to selected passing direction.

   - to pass to the next node without adding picket, press **Delete** key;
   - to pass to the previous node without adding picket, press **Backspace** key.

     To delete picket use the **Ctrl+Z** hotkeys. It is possible to delete pickets, only if history of recent operations is turned on and cancelling of these operations is allowed (see the *Undo settings* chapter in the 'General system's parameters' User Manual).

9. After passing of the last grid node the system displays the `Pathway complete` message. Click OK. Save the vector layer with acquired pickets set for further use.

To memorize a position of the current active node during passing, save the grid layer.

To continue grid nodes passing form current position, perform the following actions:

1. Load saved grid layer.
2. Choose **Vectors › Pathway › Activate** or click the button of the **Pathway mode** additional toolbar. After that a grid node which was active at the moment of the layer saving, becomes active automatically.
3. Place marker to vicinity of the active grid node.
4. Choose **Vectors › Pathway › To nearest grid node** or click the button of the **Pathway mode** additional toolbar, to move marker to the nearest node.
5. Go on to pass grid nodes.

4.1.4. Parameters of pathway mode

To specify preliminary parameters of work in pathway mode, perform the following actions:

1. Choose **Vectors › Pathway › Settings**. The **Settings** window is opened on the **Pathway** tab (see the detailed description of general system settings in the 'General system's parameters' User Manual).

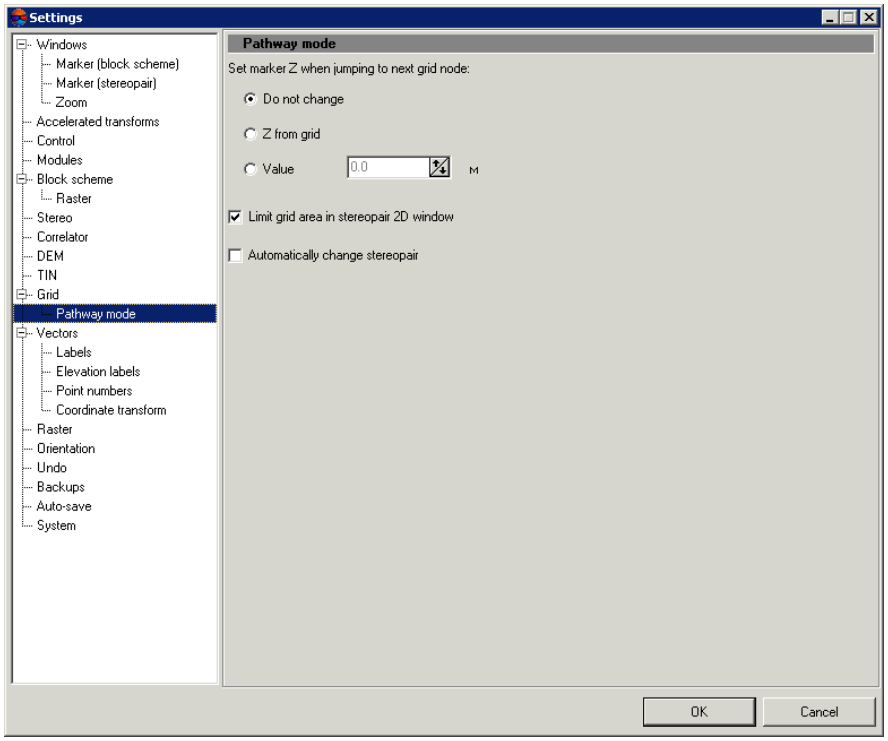


Fig. 5. Parameters of pathway mode

2. Choose one of the ways of specifying marker position by Z when pass to next grid node:

   - **Do not change** – marker position by Z is not change, i.e. Z-coordinate of marker in previous grid node is used;
   - **Z from grid** – marker position by Z is set equal to Z level of grid, defined in the **Grid properties** window;
   - **Value** – marker position by Z is set equal to the value specified in this field (in meters).

3. [optional] The **Limit grid area in stereopair 2D-window** checkbox is set on in the system by default, that allows to setup passing grid nodes in pathway mode only in borders of stereopair regardless of grid size.

It is not recommended to set the **Automatically change stereopair** checkbox on while the **Limit grid area in stereopair 2D-window** checkbox is set on, since it leads to work with incorrect data.

4. [optional] In order to pass nodes of 'global' grid with automatic stereopair change, set the **Limit grid area in stereopair 2D-window** checkbox off and set the **Automatically change stereopair** checkbox on. Besides, the **Threshold** parameter also has influence on automatic stereopair change. The parameter specifies a distance from stereopair border in percent of the whole stereopair size, and when this distance is reached during passing nodes in pathway mode the stereopair is changed to the next one in 2D-window.

4.2. Pickets automatic calculating

4.2.1. Automatic calculating of pickets

The system provides possibility to calculate pickets automatically using correlator in overlap areas of stereopair images by regular nodes grid.

All selected stereopairs should be located to adjusted part of a block (see 'Block adjustment' User Manual), otherwise, pickets calculation is not performed or is performed incorrectly. In the first case the system displays error message, in the second one – it is not possible to match calculated pickets to project coordinate system.

Acquired pickets are used as a vector base for DEM generation. Additional possibilities of pickets editing allow to obtain vector base for TIN creation and DEM generation.

The principle of automatic pickets calculation is as follows. For each selected oriented stereopair the system performs automatic pass of all grid nodes, located in overlap area of stereopair images, and tries to calculate spatial coordinates in vicinity of each grid node using correlator. In case of successful correlation the system performs accuracy control of each found point's coordinates, and after that the point is either added to vector layer as point object, or is excluded. If the system fails to calculate spatial coordinates in vicinity of some grid node, the node is skipped and the system passes to the next node.

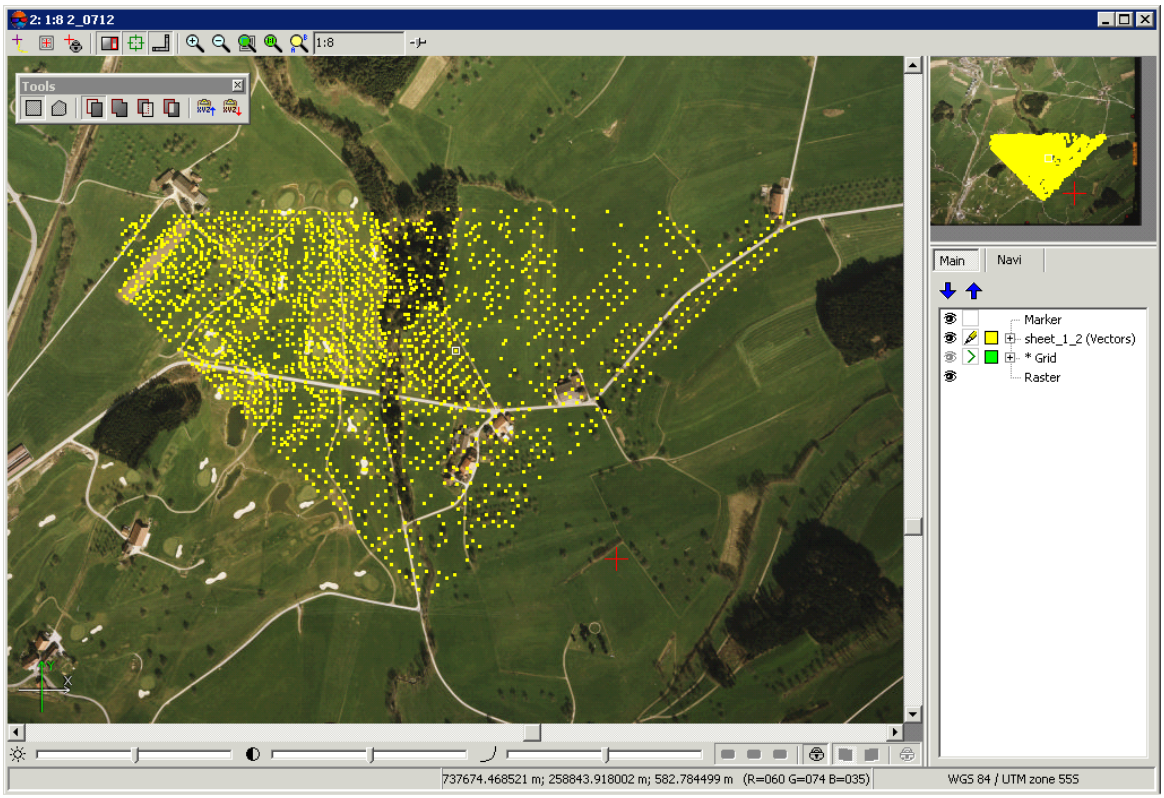


Fig. 6. Automatic pickets calculation in arbitrary area

Prior to start of automatic pickets calculation perform the following preparatory actions:

1. Define *search area*: select block stereopairs where to perform automatic pickets calculation.

2. Create regular nodes grid for selected *search area*.

3. Select **Vectors › Compute points automatically**. The **Compute points** window opens.

If a grid not was not created, it is created automatically for the entire images block and the **Grid properties** window opens, that allows to specify grid parameters.

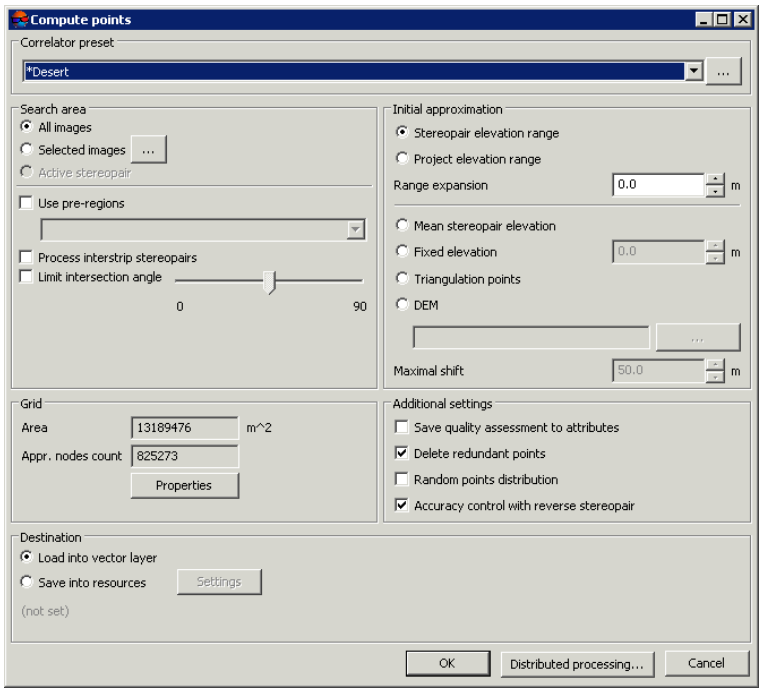


Fig. 7. The 'Compute points' window

4. In the **Correlator preset** section select in the list one of the following types of terrain:

   - **mountainous terrain**;
   - **urban area**;
   - **desert**;
   - **rural area**;
   - **rural area 2**.

   To view and edit parameters values and to setup *correlator configuration* the list, to create new or delete configurations from the list click the ... button.

5. In the **Search area** section specify a search area, which will be used for automatic pickets calculation:

   - **All images** – allows to select all images of a block;
   - **Selected images** – allows to select block images highlighted in 2D-window;

   To view and change a set of highlighted images in block 2D-window click the ... button.

- **Active stereopair** – allows to select images of a stereopair opened in active 2D-window.

All selected stereopairs should be located on adjusted part of a block (see 'Block adjustment' User Manual), otherwise, pickets calculation is not performed or is performed incorrectly. In the first case the system displays error message, in the second one – it is not possible to match calculated pickets to project coordinate system.

6. [optional] To pass each grid node only once during calculation, prior to define parameters it is necessary to create pre-regions, set the **Use pre-regions** checkbox on and select pre-regions layers in the list.

Binding of pre-regions and stereopairs is performed using `region_image_code` and `region_image_code_2` attributes.

7. [optional] Pickets calculation is performed by default for strip stereopairs. To calculate pickets for inter-strip stereopairs, formed by selected images set the **Process inter-strip stereopairs** checkbox on.

8. [optional] To filter blunders by Z on images with intersection angle of small or null value, set the **Limit intersection angle** checkbox on and define a value of minimal angle using the slider.

9. [optional] In the **Grid** section the **Area** of the grid layer in m^2, as well as **Appr. nodes count** in a grid are displayed in appropriate fields. To change a grid layer parameters click the **Properties** button.

To change grid boundaries it is necessary to close the **Compute points** window.

10. In the **Initial approximation** the method of calculation of the elevation range of points with the search for the correlator:

Initial approximation defines a value of Z coordinate of grid node, that is used to calculate initial coordinates of picket on the left and right images of a stereopair.

- **Stereopair elevation range** – the elevation range is calculated for each stereopair with heights of pickets, previously measured in the project;

- **Project elevation range** – the elevation range is obtained from the **Relief elevation** values in project properties or it is calculated with heights of pickets, previously measured in the project;

- **Mean stereopair elevation** – mean elevation of each stereopair, calculated using exterior orientation parameters, is used as initial approximation;

- **Fixed elevation** – input elevation value in meters to be used as initial approximation;
- **Triangulation points** – allows to calculate the elevation range for each stereopair from a smooth model built by project triangulation points;
- **DEM** – a value of elevation of selected DEM in point with XY coordinates of grid node is used as initial approximation.

  If grid node falls into null cell of DEM, the system uses average elevation of corresponding stereopair. To select DEM in active profile resources click the ... button.

To specify value of acceptable deviation of Z-coordinate of calculated pickets from initial approximation and input value in meters to the **Maximal shift** field.

11. [optional] There are the following options in the **Additional settings** section:

- **Save quality assessment to attributes** – allows to save quality assessment of pickets calculation as attributes of point vector object. Quality assessments are saved to the following object attributes:
  - ’corr’ (double) – correlation coefficient, number in range from specified correlation threshold up to 1;
  - ’aq’ (double) – auto-correlation sigma;
  - ’std_dev’ (double) – standard deviation;
  - ’pass’ (int) – number of passes;
  - ’disp’ (int) – dispersion value.

  To view values of attributes of pickets calculation quality select the picket and choose **Window › Object attributes**. The **Object attributes** window opens.

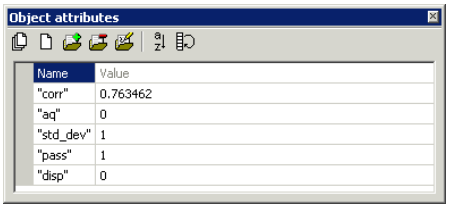


Fig. 8. Quality assessments of calculated picket

- **Delete redundant points** – allows to thin nearest pickets, acquired during passing the same grid nodes on different stereopairs;

Thinning is performed after completion of pickets calculation operation according to specified radius value, set by used correlator configuration.

When the **Save quality assessment to attributes** checkbox is on, points with minimal correlation coefficient (minimal value of '*corr*' attribute) are removed from several found nearest points.

- **Random points distribution** – allows distribute pickets randomly in area of halfsize of grid cell from grid node;
- **Accuracy control with reverse stereopair** – allows to check results of found point on a reverse stereopair; if results do not the same, point removes;

12. In the **Destination** section define parameters of loading and saving of calculated pickets:

- **Load into vector layer** – allows to load pickets into active vector layer;

If there is no active vector layer, pickets are loaded to a new vector layer.

- **Save into resources** – allows to save pickets into active profile resources without loading. Click the **Settings** button to specify parameters of pickets saving. The **Settings for saving points** window opens.

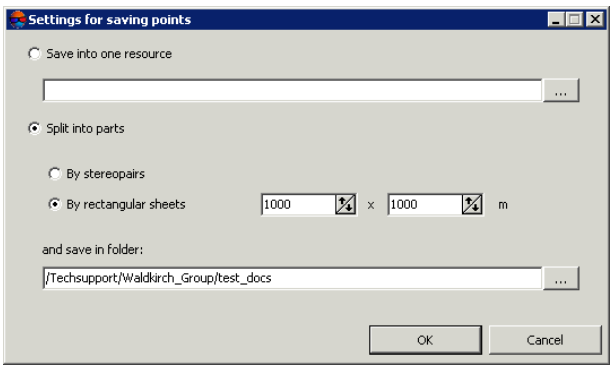


Fig. 9. The 'Settings for saving points' window

To save pickets in one vector file of active profile perform the following actions:

1. Select the **Save into one resource** option and click the ... button. The **Save** window opens.
2. Select a folder in the active profile resources and input a name of vector file in the **Resource name** field. Click OK to create a new vector file and to return to the **Settings for saving points** window.

3. The input field of the **Settings for saving points** window shows path and name of new vector file. Click OK to return to the **Compute points** window.

To save pickets in parts into several vector files of active profile perform the following actions:

1. Choose the **Split into parts** option.
2. Select one of the ways of pickets splitting – **By rectangular sheets** and input a size of rectangular area in meters or **By stereopairs**.
3. Input path to save file to the **Save in folder** field or click the ... button, to select name and path in active profile resources to save pickets in parts to corresponding vector files and click OK.
4. The input field of the **Settings for saving points** window shows path and name of new vector file. Click OK to return to the **Compute points** window.

13. Click OK. The pickets calculation operation starts.

In some cases operation of automatic pickets calculation takes a lot of time.

To setup pickets calculation in distributed mode click the Distributed processing button and specify parameters of tasks distribution.

4.2.2. Correlator configurations

Correlator configuration – is a script of pickets calculation operation, that defined a number of passes of regular grid nodes and a set of correlator parameters for each pass.

The **Correlator configuration** window is used to form a list of correlator's configurations. In order to open the window click the ... button in the **Correlator configuration** panel of the **Compute points** window.

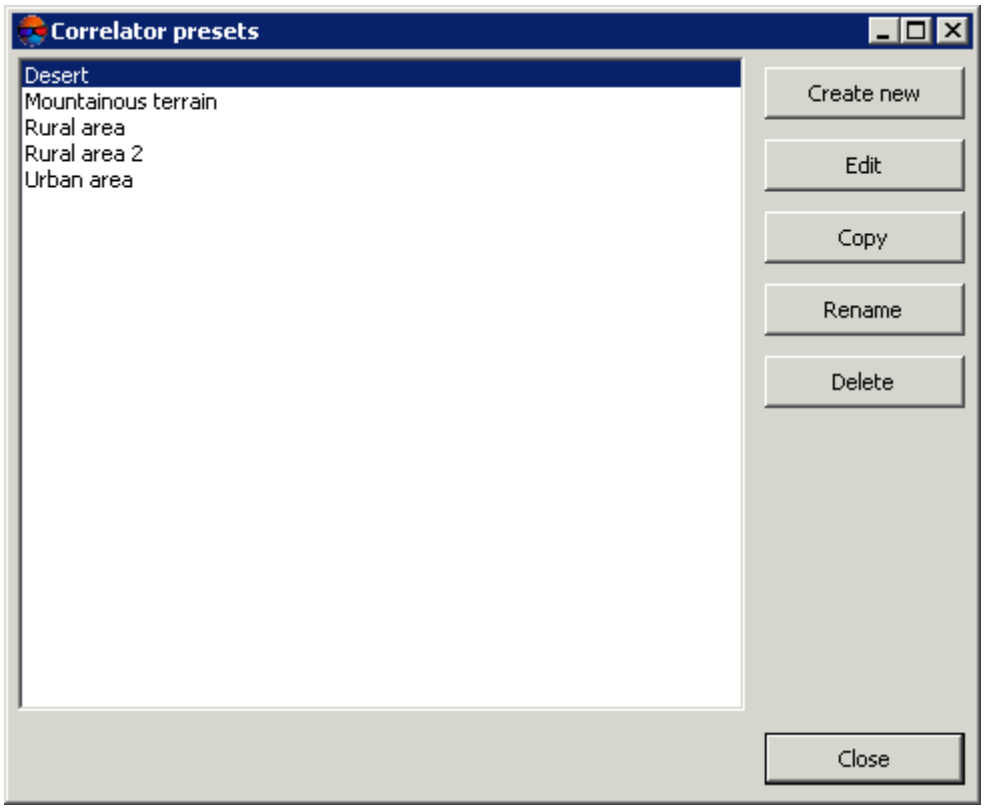


Fig. 10. The 'Correlator presets' window

The left part of the window displays a list of all created correlator configurations. Default list includes standard correlator configurations with fitted values of parameters for the following terrain types:

- *mountainous terrain* – for automatic pickets calculation on images with mainly mountainous terrain with big elevation differences;
- *urban areas* – for automatic pickets calculation on images with mainly urban type of build-up;
- *desert* – for automatic pickets calculation on images with monotonously deserted terrain with rare vegetation;
- *rural area* – for automatic pickets calculation on images with rural areas with with minor amounts of buildings and infrastructure;
- *rural area 2* – for automatic pickets calculation for 'rural' area with heterogeneous parts of relief. If there are images with big intersection angle the configuration uses a mask of larger size. Accuracy of pickets calculation is less then when using *rural area type*.

The right panel of the window contains the following buttons for managing correlator's presets:

- **Create new** – allows to specify name and parameters to create new configuration of correlator;
- **Edit** – allows to edit parameters of correlator's configuration selected in the list;
- **Copy** – allows to create a copy of selected correlator's configuration with different name, but with the same parameters;
- **Rename** – allows to rename selected correlatior's configuration;
- **Delete** – allows to delete selected correlatior's configuration;

Preset configuration couldn't be changed or removed.

- **Close** – allows to close the **Correlator presets** window.

4.2.3. Setup of correlator configuration

To view and setup correlator's configuration the system provides the **Edit preset** window, which is opened in the following cases:

- while editing of existing configuration select configuration in the list of the **Correlator presets** window and click the **Edit** button;
- while creating new configuration click the **Create new** button, input new configuration name and click OK.

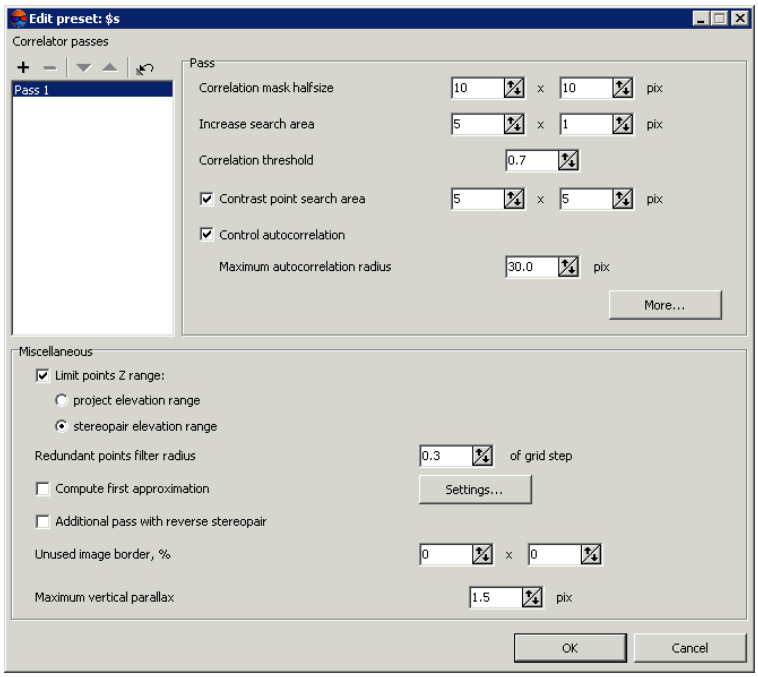


Fig. 11. Window of correlator passes parameters

Setup of correlator configuration in the **Edit preset** window includes defining of script of automatic pickets calculation.

Depending on terrain type the script could include both single and multiple passes of grid nodes, it is possible to specify particular set of correlator parameters for each of them.

For correlator configuration setup perform the following actions:

1. Create a list with required number of correlator passes.

   Passes in the list are placed according to their order of use during automatic pickets calculation.

   To form the passes list the following buttons are provided:

   - + – allows to add new pass to the end of list;
   - − – allows to delete selected pass;
   - ▼ – allows to move selected pass down the list;
   - ▲ – allows to move selected pass up the list;
   - ↶ – allows to return settings to default.

2. Select correlator pass in the list to setup its parameters.

3. In the **Pass** section setup the following main correlator parameters for a pass selected in the list:

   - **Correlation mask halfsize** – half of linear size of rectangular correlation mask in pixels by X and Y axes;

     Increasing the mask size increases recognition quality for smooth relief and decreases influence of tiny details on terrain surface, however, it decreases total amount of acquired points. Besides, during processing imagery with low-contrast areas, which correspond to sloping grounds (fields or lawns on hill slopes, for instance), the system may add points that correspond an edge of mask, rather than center.

   - The **Increase search area** parameter allows to specify a number of pixels to increase search area more than defined value of correlation mask size;

     Increasing of search area is used, when considering stereopair orientation data the estimated search area of corresponding point on the right images is insufficient (no tie points enough on high-rise objects or in stereopair corners, for instance). It is necessary to increase search area by Y in case of poor accuracy of stereopair relative orientation.

   - **Correlation threshold** – minimal acceptable value of correlation coefficient;

     Value of correlation threshold depends significantly both on material and other parameters. For correct work of correlation algorithm it is recommended to setup the value not less than 0.7.

   - **Contrast point search area** – a size of area around grid node, where the system performs search for a point with maximal contrast prior to correlation operation;

     It is recommended to increase this value for images with well-defined contours of objects, and to decrease it for low-contrast (fuzzy) areas, where there is high probability of correlator errors.

   - **Control autocorrelation** and **Maximum autocorrelation radius** – is used to control point's autocorrelation – degree of point's uniqueness in some its vicinity on the left image.

     The more the autocorrelation radius, the less distinctive is the point and the more probable is incorrect matching of the point with the right image, including cases with high correlation coefficient.

   The **More** button is used to setup additional parameters of selected pass.

4. In the **Miscellaneous** section setup the following additional parameters of correlator's pass:

- **Limit points Z range** – allows to specify one of filtering criteria by Z-range, which is applied to each point calculated using calculator:
  - **project elevation range**, defined by user in project properties;
  - **stereopair elevation range**, calculated during images block adjustment.
- **Redundant points filter radius** – is used to filter nearest pickets, acquired during passing the same grid nodes on different stereopairs;

  Pickets thinning is performed after completing of pickets calculation operation, considering specified value of the radius in meters.

- **Compute first approximation** – allows to use results of preliminary pickets calculation, which parameters are setup using the **Settings** button;

  It is recommended to perform preliminary calculation for automatic search for pickets on satellite scanner images.

- **Additional pass with reverse stereopair** – allows to use additional pass during pickets calculation, when the system performs search for matching points on the left image for points on the right image (during main pass the system performs points search on the right image for points on the left one);
- **Unused image border** (in percent) – allows to not consider specified percent of image edges during pickets calculation;
- **Maximum vertical parallax** – allows to specify a limit for points filtering by vertical parallax value (in pixels).

Point acquired as a result of correlation has 4 coordinates – (XL, YL) on the left image and (XR, YR) on the right image. To calculate vertical parallax the system performs transformation of images coordinate system into project coordinate system and vice versa. As a result two pairs of new coordinates are acquired (XL_2, YL_2) and (XR_2, YR_2). In this case parallax is calculated as $Y_{PAR} = (YR - YL) - (YR_2 - YL_2)$. Points with the value more than specified Y_{PAR} are rejected. This parameter should be selected experimentally, especially for scanner survey projects.

5. Click OK to complete setup of correlator configuration parameters.

4.2.4. Additional correlator's parameters

To setup additional parameters of correlator for selected pass in the **Edit preset** window there is the **More** button, that allows to open the **Additional settings:** window **Pass #**.

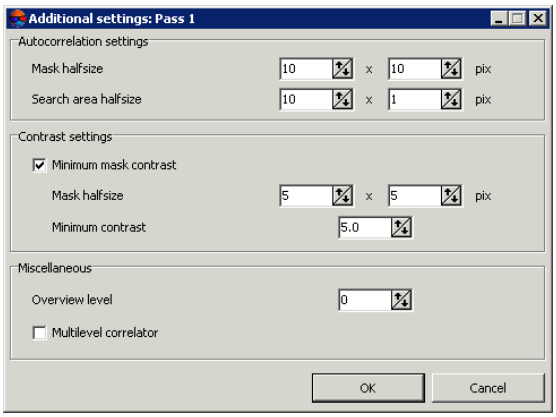


Fig. 12. Additional parameters of selected pass

The **Autocorrelation settings** section allows to specify additional parameters of auto-correlation:

- **Mask halfsize** – half of linear size of mask for auto-correlation in pixels by X and Y axes. Default value – 10x10 pixels;
- **Search area halfsize** – a halfsize of area, where auto-correaltion surface is created.

The **Contrast settings** section allows to consider contrast for low contrast images. Set the **Minimum mask contrast** checkbox on and specify the following parameters:

- **Mask halfsize** – a halfsize of area around a point, where the system performs contrast calculation;
- **Minimum contrast** – value lies in range from 0 to 100.

The **Overview level** parameter allows to perform correlation between thinned images and is used in the following cases:

’0’ value corresponds to initial image, in other cases that system uses the level of the pyramid with a specified number.

- for film data with coarse grain, which prevent from reliable operation of correlator;
- on relief surface like fields, that in initial detalization have occasional irregular items (grass, small bushes), or regular pattern (new-ploughed field). When zoom out such a terrain surface, the textures comparable on the left and right images appear;
- to exclude small objects on relief (bushes, separate trees, buildings). In this case, the thinning together with increasing of mask size leads to the fact that even if the center of the mask falls onto the protruding object, the resulting Z-coordinate of picket corresponds to the surrounding terrain elevation.

As the passes of the same calculation process, it is advisable to use a correlator configuration differing by no more than one level of the pyramid (for example, 0 и 1; 2 и 3). Otherwise, in areas of fine detail (in particular, buildings), it is possible to receive the 'chess' grids of alternating points at the height of the object, and at the height of the relief.

The **Multilevel correlator** checkbox allows to use different levels of pyramid for correlation during search for corresponding points.

4.2.5. Parameters of first approximation calculation

The system provides possibility to use first approximation of digital terrain model during correlator passes. Such DTM is a TIN, generated using thinned pickets set.

DTM of first approximation allows to significantly reduce the search area for each point of the main operation, i.e. to reduce the total search time and reduce the probability of erroneous points occurrence.

In correlator's preset configurations first approximation calculation is used for *'mountainous terrain'* configuration.

To use and configure calculation parameters of first approximation DTM, perform the following actions:

1. In the **Edit** preset window set the **Compute first approximation** checkbox on.
2. Click the **Settings** button. The **First approximation for:** window opens **'configuration name'**.

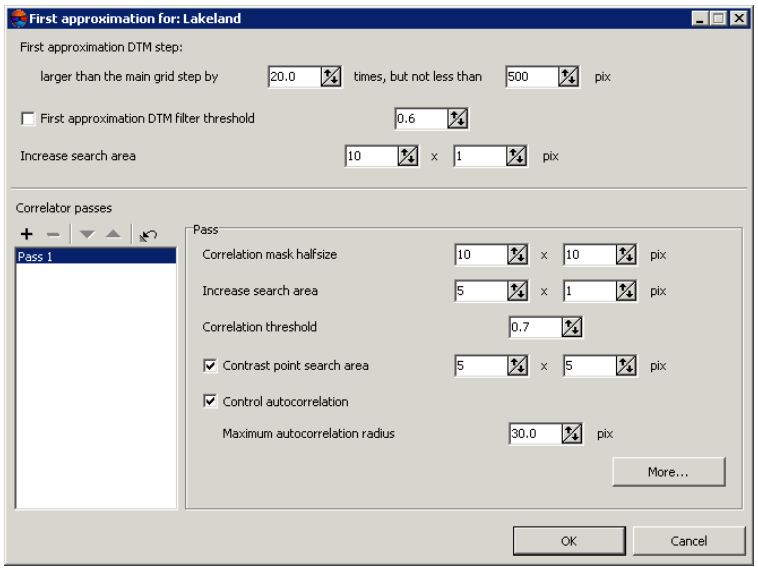


Fig. 13. Parameters of first approximation calculation

3. Specify the **First approximation DTM step** to define a step of DTM to be created.

4. [optional] To use threshold filtering during DTM creation set the **First approximation DTM filter threshold** checkbox on. Specify a value that corresponds to threshold of filtering of intermediate TIN by minimum angle of inclination of the normal, given to a range of (0;1).

5. In the **Increase search area** fields input a number of pixels for additional increasing of search area and to make it more than values, specified in the main parameters of correlator passes.

6. The **Correlator passes** list and parameters in the **Pass** section allow to define correlator configuration for preliminary pickets calculation (see a description of the main parameters in the Section 4.2.3).

   The **More** button allows to setup additional parameters of selected pass.

7. Click OK to return to the **Edit preset** window.

4.2.6. Pickets calculation in distributed processing mode

To calculate pickets in distributed processing mode, perform the following actions:

1. Change settings and run the distributed processing server/client (see the '*Distributed processing*' chapter in the 'General information about system' User Manual).

2. Select **Vectors › Compute points automatically**. The **Compute points** window opens.

   If a grid not was not created, it is created automatically for the entire images block and the **Grid properties** window opens, that allows to specify grid parameters.

3. Setup parameters of automatic pickets calculation.

4. Click the **Distributed processing** button. The **Compute points Distributed Processing** window opens.

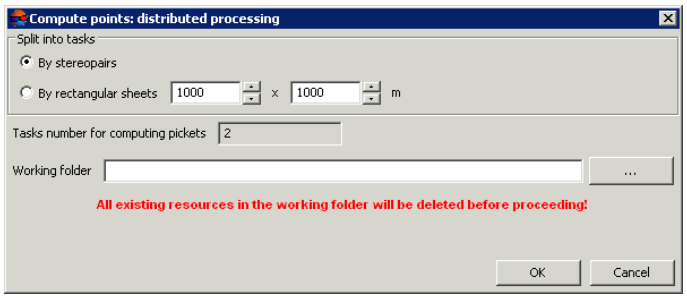


Fig. 14. Parameters of distributed processing of pickets calculation

5. In the **Split into tasks** section select a way of splitting of processing operation into tasks:

   - **By stereopairs** – allows to process each stereopair in a separate task;

     Since tasks are executed independently from each other, the system doesn't remove points, falling into the same grid nodes from different stereopairs (for example, during filtering of nearest points).

   - **By rectangular sheets** – it is necessary to specify in the fields a size of rectangles, into which the entire search area is split (in meters). Filtering of nearest pickets is not performed in this case. The result of each task processing is saved to the file with name like `Sheet_X_Y.x-data`.

     Rectangle, circumscribing grid boundaries in project coordinates system, is split by rectangular sheets of a size specified in the fields. Each sheet includes images parts from all stereopairs, into which it falls.

   If stereopairs are processed in separate tasks, after that it is necessary to perform manual data correction. Calculated pickets are saved in resources, named by stereopairs names.

   The **Tasks number for computing points** field displays calculated number of tasks depending on selected way of workflow split and/or size of specified areas.

   By default По умолчанию задается вложенная папка в папке *Data* текущего проекта.

   It is not possible to define number of tasks manually.

6. In the **Working folder** section click the ... button and select **empty** folder in active profile resources to save output DEM.

   Before operation is performed all data from selected folder is removed. It is strongly not recommended to specify a project folder as a working folder, especially when there are no saved project backups.

7. Click OK. Distributed processing tasks are created and the system shows a message about number of created tasks.

   When using distributed pickets calculation, the result of each task is saved to separate file with `tsk` extension in specified **Working folder** of a project.

4.3. Pickets editing

The system provides possibility of editing of single or multiple selected pickets. Since pickets calculated automatically are points, all operations of points editing are applicable to them.

To select vector objects manually in active vector layer the system provides the following features:

- mouse double click or the **S** key to highlight single polyline/polygon;
- mouse click to highlight point/vertex (place marker in point's vicinity in advance);
- selection mode **Rectangle** to select vector objects inside a rectangle;
- selection mode **Polygon** to select vector objects inside arbitrary polygon.

To highlight objects on active vector layer *inside a rectangle* choose **Edit › Group selection › Rectangle** or click the button of the **Tools** toolbar. Press and hold the **Shift** key and drag a rectangle by mouse.

To highlight objects *inside arbitrary polygon* choose **Edit › Group selection › Polygon** or click the button of the **Tools** toolbar. Press and hold the **Shift** key. Then make mouse click to create the first polygon vertex and a 'rubber line' directed to current cursor position. Input subsequent vertices by mouse clicks. To complete objects selection by polygon use mouse double click. To break off selection, press the **Esc** key.

For quick switching between **Polygon** and **Rectangle** selection modes there is context menu opened by the **Shift+F10** hotkeys.

To select all objects of active vector layer choose **Edit › Select all** or use the **Ctrl+A** hotkeys.

See detailed description of selection tools and modes in the 'Vector objects selection' chapter of the 'Vectorization' User Manual.

To add a new picket to active vector layer, perform the following actions:

1. Choose **Edit › Vectors create mode › Points** (**P**) or click the button of the **Vectors** additional toolbar to turn on pickets creation mode.
2. Place marker to selected point on image in 2D-window.
3. Press the **Insert** key to add a point without correlator on specified elevation.

   The system also allows to place marker on a model surface automatically using correlator. The **Space** key is used to do this.

If the correlator failed to work the **Status** panel displays the `Bad point` message and the system produces warning audio signal (see the 'General system's parameters' User Manual).

Besides the system provides possibility to delete pickets, located nearer from linear vector objects than specified distance (see the 'Vectorization' User Manual).

In this document all points not lying on the relief surface (average smoothed terrain surface), are called *surges*.

The system provides deletion, correction or recognizing pints (pickets) which have fallen on buildings, trees, cars, in pits, etc. during automatic pickets calculation, and during filtering of accident surges.

To edit automatically calculated pickets the system provides the following filters:

- Pickets thinning – thinning out of densely located points with specified level of thinning;
- Filter by Z-range – filtering points and vertices of polylines/polygons, which Z-coordinate is out of specified range;
- Median Z filter – filtering points and vertices of polylines/polygons using mask of specified size;
- Filter adjacent point objects – filtering of closely located points (located nearer than specified distance);
- Buildings and vegetation filter – filtering points, which have fallen on tall objects (buildings, trees) or in pits to obtain layer of points, describing only terrain relief;
- Linear objects filtering – filtering polylines/polygons, that linear size (coordinates range by one of axes) is more or less of specified value;
- Surface objects filter – filtering points, which have fallen on particular tall objects or pits of specific size.

The system provides possibility of vector objects topology control, i.e. search for duplicated pickets or pickets coinciding with vector objects vertices. For this the **Vectors › Topology › Verify topology** menu item is used (see the chapter 'Topology verifying' of the 'Vectorization' User Manual).

4.4. Filtering

4.4.1. Pickets thinning

The system provides possibility of removing of some part of automatically calculated pickets, located densely or close to each other. For this there is pickets thinning feature.

To perform pickets thinning perform the following actions:

1. Select **Vectors › Filter › Thinning**. The **Pickets thinning** window opens.

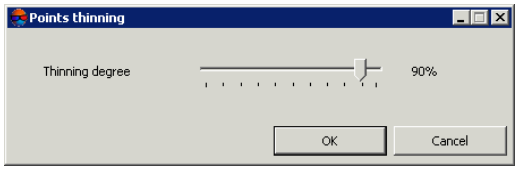


Fig. 15. Pickets thinning parameters

2. Specify degree of pickets thinning out in percent using the slider.

3. Click OK. After that the system removes specified percent of points, in order of their influence on quality of relief representation (the least significant points are removed first).

4.4.2. Pickets filtering by Z-range

The system provides possibility to remove points and vertices of polylines/polygons, which Z-coordinate falls outside specified range.

To perform objects filtering of active vector layer by elevations range, execute the following actions:

1. Select **Vectors › Filter › Filter by Z-range**. The **Z-range filter** window opens.

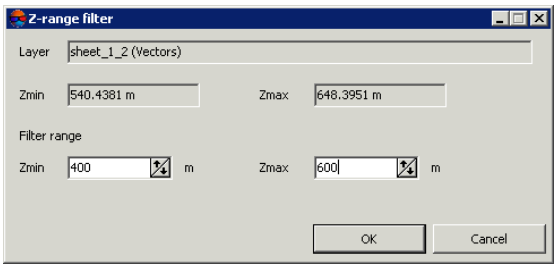


Fig. 16. Parameters of pickets filtering by Z

   The **Layer** field displays a name of selected active vector layer, the **Zmin** and **Zmax** fields show terrain elevation difference in meters, calculated by all layer objects.

2. The **Filter range** section by default shows values of calculated terrain elevation difference. Specify maximal and minimal Z values in meters to consider for points filtering.

3. Click OK. Filtration removes all points and vertices of polylines/polygons, which Z-coordinate falls outside specified range. When the filtering operation is completed the system displays information message about number of deleted points.

Check and filtering by Z-range is also possible during automatic pickets calculation.

4.4.3. Median pickets filtering by Z

The system provides possibility of median filtering to remove single sharp spikes in presence of smooth relief.

The *median filtering* contains the following sequence of actions:

1. Sequential scanning of area with vector objects by window-mask. Scanning step is specified as window-mask halfsize.

2. Deleting points which elevation is out of range.

The range is calculated as average level of vectors elevation that fall into mask. When calculating the range the system considers specified deviation from average level.

To execute median filtering perform the following actions:

1. Select **Vectors › Filter › Median Z-filter**. The **Median Z filter** window opens.

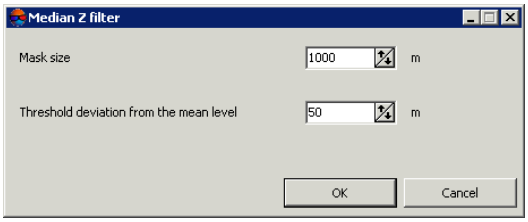


Fig. 17. Parameters of median pickets filtering

2. In the **Mask size** field specify a side size of scanning square window-mask in meters.

3. In the **Threshold deviation from the mean level** field specify deviation by Z in meters from mean level of objects elevation in the mask.

4. Click OK. Filtration removes all points and vertices of polylines/polygons, which Z-coordinate falls outside specified range. When the filtering operation is completed the system displays information message about number of deleted points.

4.4.4. Filter of adjacent point objects

Filter of adjacent point objects allows to remove pickets, in which vicinity there are other objects located closer specified distance.

To filter adjacent pickets on active layer perform the following actions:

1. Select **Vectors › Filter › Filter adjacent point objects**. The **Filter adjacent point objects** window opens.

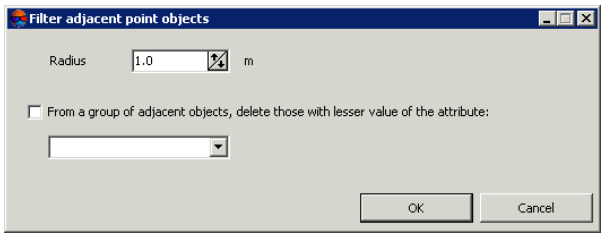


Fig. 18. Parameters of adjacent point objects filter

2. Specify the **Radius** – maximal distance in meters to each picket, and if a pickets falls closer that the radius value, the pickets is deleted.

3. [optional] In order to delete from two pickets just one which contains the smallest value of one attribute (for example, if pickets vector layer was created in automatic pickets calculation mode with preserving of quality assessment in attributes), set the **From a group of adjacent objects, delete those with lesser value of the attribute** checkbox on and select in the list an attribute to be considered during pickets removal.

4. Click OK. Filtration removes pickets, between which the distance is less than specified. When the filtering operation is completed the system displays information message about number of deleted points.

4.4.5. Buildings and vegetation filter

The system provides deletion, correction or recognizing pickets which have fallen on buildings, trees, cars, in pits during automatic pickets calculation, and during filtering of accident surges. For that filter of buildings and vegetation id used. After its work only pickets that describe terrain relief are remain in the system.

In this document all points not lying on the relief surface (average smoothed terrain surface), are called *surges*.

Filter of buildings and vegetation allows to apply step-by-step points filtering using particular script, i.e. to apply pickets filtering in multiple passes with different parameter sets.

It is possible to develop customized script of filter workflow or to use default script. Creating script of buildings and vegetation filtering means specifying of passes set, order of their passing and setup the parameters for each pass.

Step-by-step filtering allows to achieve optimal results of objects filtering on any type of terrain to acquire pickets, that correctly describe terrain relief, and that could be used to generate high quality DEM.

By default, the system uses standard script that includes three passes in the following order:

1. **Basic** – main pass with parameters configured for filtering points not lying on the relief surface (surges). On this step the system rejects majority of points on buildings, and also correlators blunders (sharp spikes).

2. **Additional** – additional pass with parameters configured for searching surges, missed during basic step.

3. **Detailed** (off by default) – the pass with parameters configured for filtering points located on low objects, such as small buildings, cars, and so on.

After filtering the system may perform the following actions with points found (surges):

- rejecting surges from initial pickets layer;
- correction of surges found on initial pickets layer, that means editing of surges Z-co-ordinates;
- search for surges to analyse them with saving of found points to new vector layers without changing initial pickets layer.

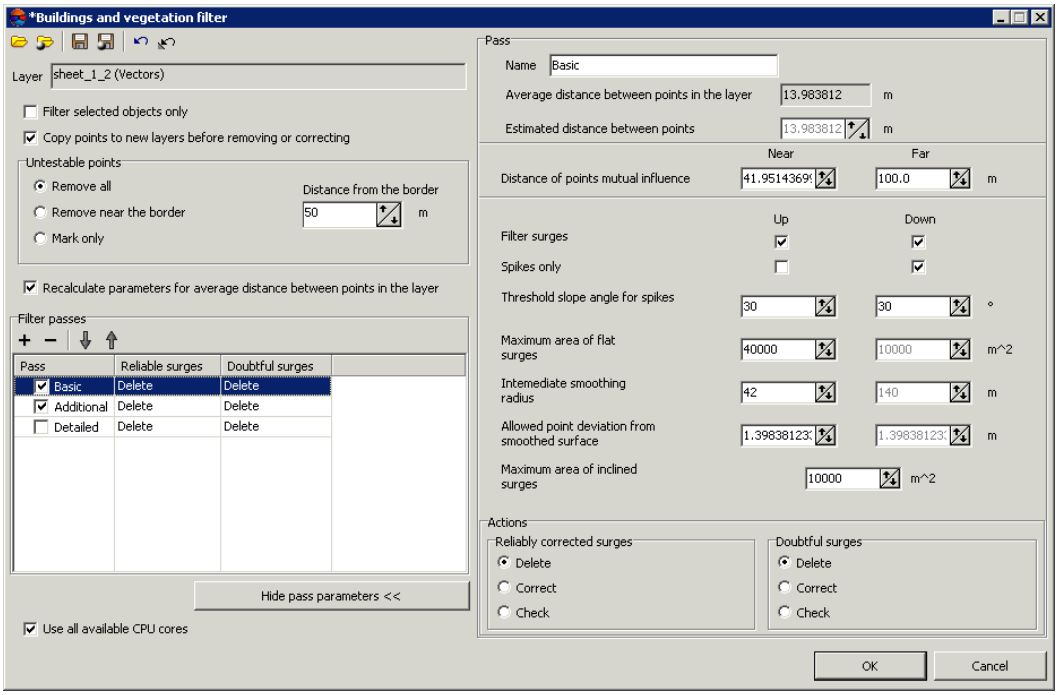


Fig. 19. Parameters of buildings and vegetation filter

The **Buildings and vegetation filter** window allows to specify filtering parameters and setup a number of filter passes. The **Layer** field displays a name of active vector layer. The window contains a standard toolbar.

Buttons	Function
	allows to load a script of filter work from `*.x-filter` resource located out of active profile resources
	allows to load a script of filter work from `*.x-filter` resource located in active profile resources
	allows to save a current script of filter work to `*.x-filter` resource located out of active profile resources
	allows to save a current script of filter work to `*.x-filter` resource located in active profile resources
	allows to cancel all changes made to a script
	allows to go back to a standard script that includes two passes with default settings (regardless of which script has been loaded)

To filter objects on surface perform the following actions:

1. Load pickets for filtration or make active a pickets layer.

   To avoid loss of data, it is recommended to use a copy of initial layer during filtering.

   To create a copy select **Vectors › Save as** and specify new name of pickets file.

   If a layer contains linear or area vector objects the filter may not work properly.

2. Select **Vectors › Filter › Buildings and vegetation filter**. The **Buildings and vegetation filter** window opens.

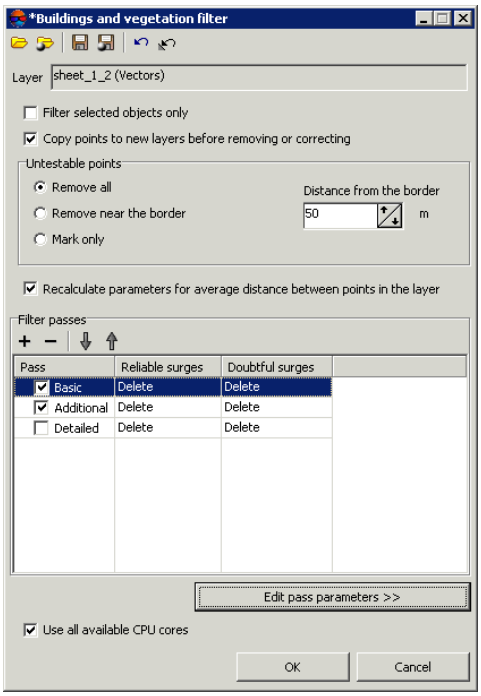


Fig. 20. Parameters of buildings and vegetation filter

3. [optional] In order to use filtering for selected objects (points group), set the **Filter selected objects only** checkbox on. Otherwise, filter is applied to all points of the layer.

   To filter a group of points, select it by mouse in 2D-window prior to start configuring the **buildings and vegetation filter**.

   When using filter for selected objects only, it is not recommended to set the **Remove near the border** option in the **Untestable points** section.

4. [optional] To increase the filtering operation performance set the **Copy points to new layers before removing or correcting** checkbox off. By default, the system saves removed or corrected pickets in new vector layers used for analysis of filtering results. These layers could be used to restore basic pickets layer (using operation of layers merging), if the copy of initial pickets layer was not created prior to filtering start.

   If the **Copy points to new layers before removing or correcting** checkbox is off, it is highly not recommended to apply filtering to initial pickets layer, since it is impossible to restore the initial layer after filtering.

5. In the **Untestable points** section specify one of the following actions with untestable pickets found after filtering:

Some points can be *untestable* – the points, in which vicinity there is not enough 'neighbour' points. For example, points located on the edge of the whole layer or in areas of low density of points (fields, forests, water bodies).

- **Remove all** – allows to remove all found untested pickets;
- **Remove near the border** – allows to set a value of distance in meters from the edge of the whole vector layer, at which all untestable pickets will be removed. The **Distance from the border** field is used to do this;
- **Mark only** – allows to save untested pickets to separate vector layer for further analysis.

6. [optional] Values of average distance between pickets in active layer calculates automatically. Also the **estimated distance between points** is equal to average distance by default. Set the **Recalculate parameters for average distance between points in the layer** checkbox off to set the **estimated distance between points** manually.

7. Specify a number of filter passes:

   - by default it is used two passes with standard parameters – **Basic** and **Additional**. Set off checkboxes to change set of passes;
   - the + button allows to add a new pass of filter;
   - the − button allows to remove selected pass of filter;
   - the ⇩ and ⇧ buttons allow to move selected pass up/down.

   The **Filter passes** section contains a table of passes and buttons used to change set of passes and their order with the following columns:

   - **Pass** – displays pass name and checkbox used to include a pass to script or to exclude it from filter workflow script;
   - **Reliable surges** – displays action to be applied to well-defined surges found, that are completely comply with specified filtering criteria;
   - **Doubtful surges** – displays action to be applied to ambiguous surges found, in which vicinity there is no data enough for analysis.

   It is necessary to select an action to be applied to reliable and doubtful surges during configuring of pass parameters.

8. Select a pass name and click the **Edit pass parameters> >** button. The **Pass** section is opened and allows to display and configure the following parameters of filter pass:

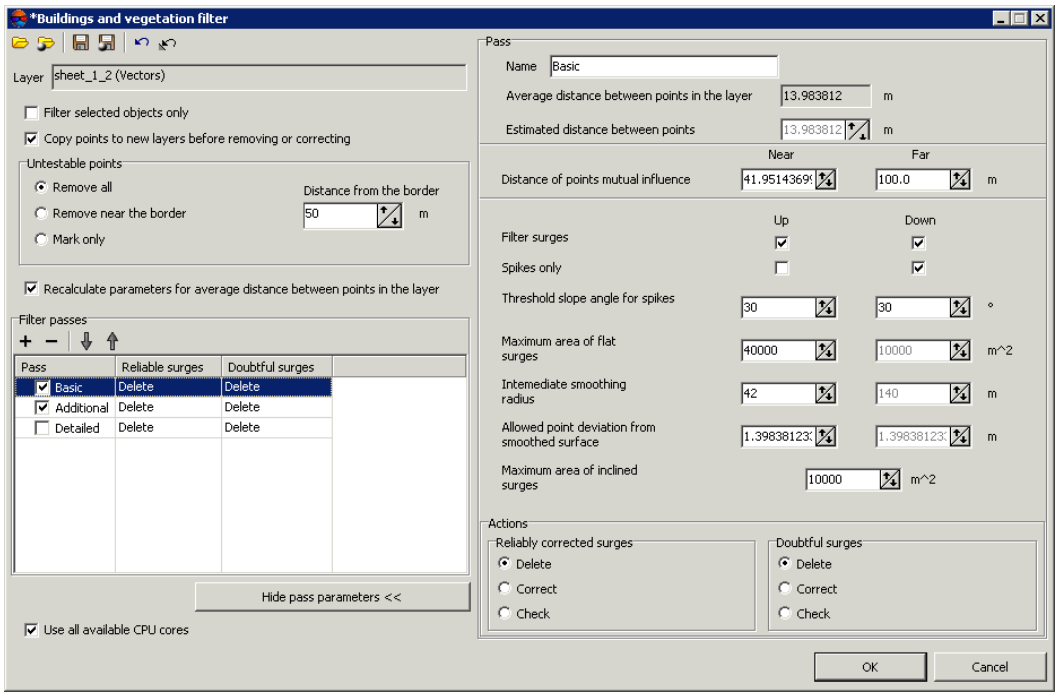


Fig. 21. Parameters of buildings and vegetation filter

- [optional] to change name of pass, input a new one in the **Name** field;
- the **Average distance between points in the layer** (in meters) and **Estimated distance between points** (in meters) after filtering are displayed in fields;
- specify **Near** and **Far distance of points mutual influence** in meters to define the circle radius. In this area points are checking for errors;

  It is recommended to set the near distance (minimum circle radius) value as 3-5 values of **Average distance between points in the layer** parameter.

- specify the following filtering parameters **Up** (above surface) and/or **Down** (below surface):
  - **Filter surges** – allows to select pickets to be filtered: pickets on surface (high objects) and/or pickets below surface (pits);

    For filter operating it is necessary to set at least one checkbox on.

  - **Spikes only** – is used to reject just sharp spikes above/below surface, that are defined by values of the **Threshold slope angle for spikes** and **Distance of points** parameters. The rest of parameters are not considered;

In an angle between three points exceeds a value of the **Threshold slope angle for spikes** parameter, then the surge is called *spike*.

- **Threshold slope angle for spikes** – allows to specify angle of slope in relation to selected measuring plane (above and/or below surface) to define sharp spikes;
- **Maximum area of flat surges** – allows to define maximal area of plane surges (above/below surface) – points groups, that form smooth surfaces and are distant from some plane of neighbour points. Generally, the points are located on buildings roofs, and are lying on the same plane;

The filter is not applied to plane surface, which area exceeds the specified value.

- **Intermediate smoothing radius** – allows to specify a sphere radius (above/below surface), that defines a level of intermediate smoothing of surface;
- **Allowed point deviation from from smoothed surface** – allows to specify criterion, according which the filter is applied to all pickets with elevations that differ from smoothed surface more then on specified value;
- **Maximum area of flat surges** – allows to define maximal area of plane surges (above/below surface) – points groups, that form smooth surfaces and are distant from some plane of neighbour points.

The button allows to go back to a standard script that includes two passes with all default settings (regardless of which script has been loaded).

9. In the **Actions** section specify actions for reliable and doubtful surges found after any step of filtering:

   - **Delete** – deleting of pickets found from base pickets layer;
   - **Correct** – editing in base layer Z-coordinates of points found;
   - **Check** – saving of found pickets in a new layer without change of base layer.

   Reliable corrected surges – found well-defined surges that satisfy completely to all specified filtering criteria.

   Doubtful surges – ambiguous surges in which vicinity there is no data enough to analyse filtering operation possibility.

   If the **Delete** or **Correct** option is selected, the system allows to edit base pickets layer.

In order to save deleted or corrected pickets to new layers it is recommended to set the **Copy points to new layers before removing or correcting** checkbox on.

If the **Check** option is selected the initial pickets layer remains unchanged, and found reliable or doubtful surges are copied to a new layer *Surges*.

10. [optional] To use all cores of workstation CPU for calculation operation, the **Use all available CPU cores** checkbox is set on in the system by default. It is necessary to set the checkbox off to use only one core.

11. Click OK. The system starts DEM filtering operation. When the filtering operation is completed the system displays information message about number of used basic points and filtered pickets.

There are the following recommendations concerning use of buildings and vegetation filter:

- it is recommended to use step-by-step filtering (in multiple passes with different parameter sets);
- the passes should be formed in ascending order of the following parameters: distances of mutual influence, radius of intermediate smoothing, and decreasing of allowed deviation of picket from smoothed surface;
- value of intermediate smoothing radius directly describes terrain relief features, that it why it is necessary to specify the radius value not exceeding 1500 meters;
- it is recommended to perform preliminary analysis of maximal area of flat surges on given territory (to measure it in a stereopair window) and then to compare it with area, calculated automatically for passes. If the measured area is bigger than calculated one, it is necessary to increase it;
- it is highly not recommended to specify a distance of mutual influence of pickets exceeding a step between pickets in more than 20 times, since it leads to considerable slowdown of filtering operation.

4.4.6. Filtering of linear objects

The system provides possibility to remove vector objects, linear size of which does not satisfy the specified value.

To use linear filtering perform the following actions:

1. Define linear size of objects to be deleted.

2. Select **Vectors › Filter › Selective delete objects**. The **Delete lines** window opens.

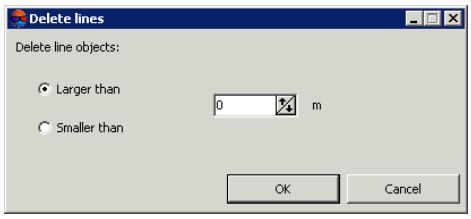


Fig. 22. Parameters of linear objects filtering

3. Select filtering method:

   - **Larger than** – used to delete objects, linear size of which exceeds the specified value (in meters);
   - **Smaller than** – used to delete objects, linear size of which is less than specified value (in meters).

4. Click OK. Filtration removes all linear objects with size that is more or less than specified. When the filtering operation is completed the system displays information message about number of deleted linear objects.

4.4.7. Surface objects filter

The system provides possibility to correct blunders of correlation, obtained during automatic pickets calculation. The filtering allows to delete points located on terrain surface with specified typical size by XY and Z.

This filtering type is not used for objects of completely built-up area, forest belts and similar objects.

The principle of filtering is to estimate the location and size of the contours created using TIN, which was generated with help of initial pickets. A surge is a group of contours that is not greater than specified size by XY and with elevations in specified range.

Surface objects filter is recommended to use to eliminate sharp spikes. For more fine filtering it is recommended to use this filter along with buildings and vegetation filter.

To apply filter of objects on surface perform the following actions:

1. Select **Vectors › Filter › Surface objects filter**. The **Surface objects filter** window opens.

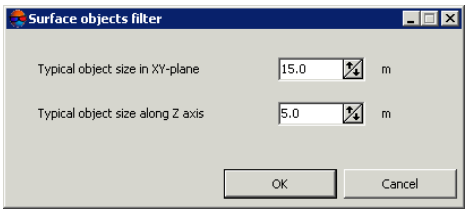


Fig. 23. Parameters of surface objects filter

2. Specify the **Typical object size in XY-plane** and **Typical object size along Z axis** in corresponding fields in meters.

3. Click OK to remove pickets found.

5. Triangulation irregular network (TIN)

5.1. The ’TIN’ menu

The **TIN** menu contains standard menu items used to load and save TIN layers, as well as to perform different operations on creation, editing and accuracy control of TIN building.

Table 2. Brief description of the ’TIN’ menu

The TIN menu	Function
Create (**Ctrl+N, T**)	allows to create TIN using loaded base layers
Compute border	allows to create TIN border in automatic mode
Rebuild	allows to rebuild TIN after editing of base layer objects
Check against adjustment points	is used to control TIN creation accuracy by triangulation points
Check against vector objects	is used to control TIN creation accuracy by vector objects, which were not used during TIN creation
Verify topology...	is used to check topology of TIN creation
The area of a polygon on the surface	allows to calculate TIN surface area within a polygon
Calculate area	allows to calculate TIN projection area on plane and area of TIN 3D surface
Statistic	is used to view statistic information about TIN
Simplify	is used to filtering of TIN triangles nodes using filtering of point vector objects in base layers, used for TIN creation
Peak filter	is used to peaks filtering – searching for TIN nodes, which Z value differs from adjacent ones significantly
Filter by Z-range	is used to edit TIN using Z-filter
Interpolate	Allows to interpolate TIN to smooth DTM in order to build or enhance contour lines

The TIN menu	Function
Embed objects	Allows to embed vector objects layer to created TIN
Export	Contains menu items to export results of TIN creation to DXF and CSV formats

5.2. General information

Triangulation Irregular Network (*TIN*) – one of the models of spatially coordinated data, used during designing of digital terrain model as elevation points in nodes of irregular network of triangles, that corresponds to Delaunay triangulation;

Delaunay triangulation – is a triangle polygonal network, formed on a set of discretely placed points connected to each other by disjoint straight line segments in such a way, that a circle circumscribed around each triangle does not contain points of initial set inside.

The system also provides a possibility to build *quasi-contours* – isolines with specified step, which cross sides of TIN triangles. Quasi-contours are rebuilt automatically during editing of base layers containing vector objects. Quasi-contours are used to provide additional quality control of TIN generating and to reveal blunders.

During saving or loading TIN all base vector layers used for TIN creation are saved/loaded.

During TIN creation it is recommended to use breaklines – vector polylines along specific landforms (for example, watershed lines and valley-lines), as well as linear objects of human origin (for example, road-sides).

To perform TIN creation the system provides the following workflow:

1. [optional] Pre-regions building.
2. Defining of set of base vector layers.
3. TIN borders creation.
4. Building TIN.

The system provides the following features to work with TIN:

- different display ways;
- area calculation;
- TIN re-build after base layers change;
- filtering of base layers, used for TIN creation;
- accuracy control of TIN creation;

- export TIN to exchange formats.

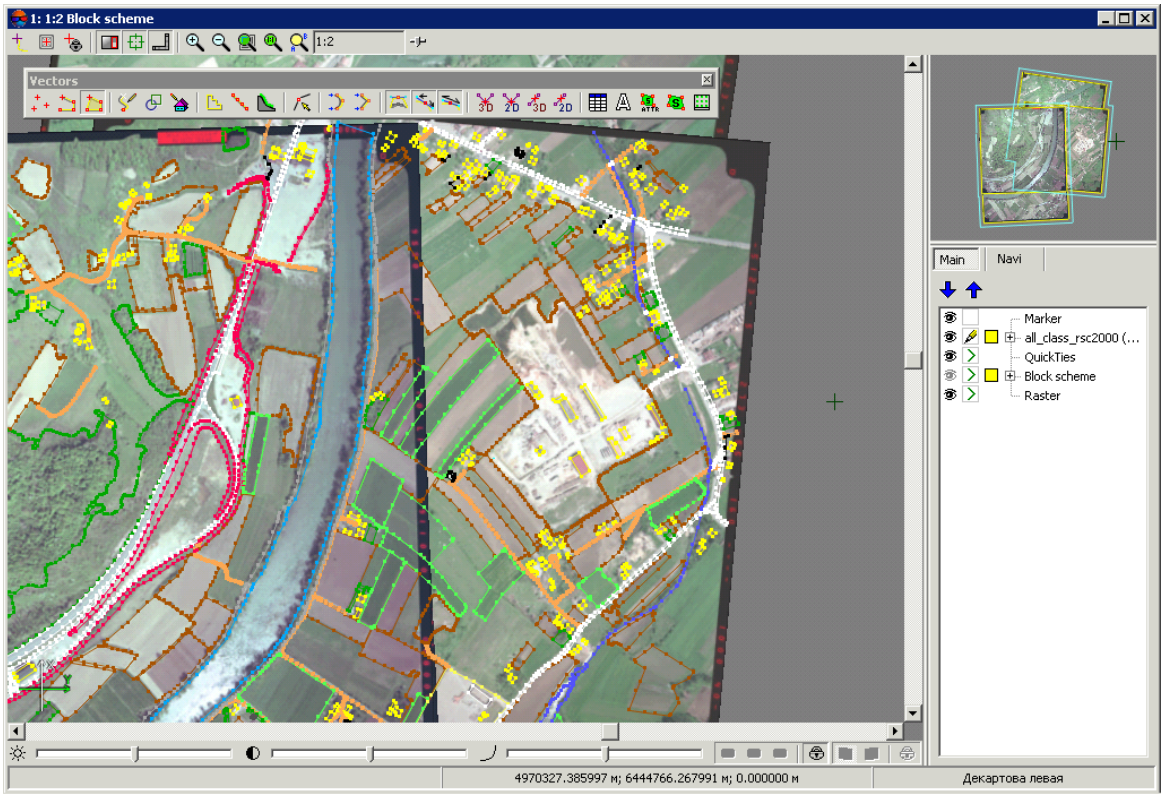


Fig. 24. Source data for TIN creation

5.3. TIN creation

5.3.1. Creation pre-regions for a block

The system provides possibility to create block pre-regions by images frames or by stereopair frames to allow mutual project processing during vectorization. Pre-regions created by stereopairs also allows to considerably speed up automatic pickets calculation and generation of dense relief model due to reducing of number of block images overlap areas.

Pre-regions – vector layer which consists of polygons, drawn by images block.

Perform the following actions to create pre-regions:

1. Select **Block › Build pre-regions**. The **Create pre-regions** window opens.

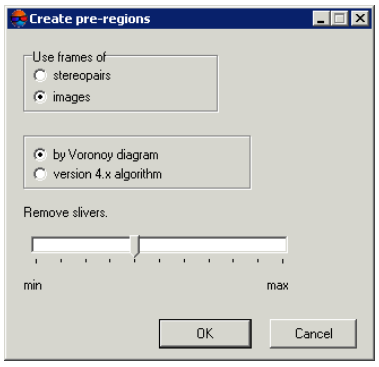


Fig. 25. Parameters of pre-regions creation

2. In the **Use frames of** section select one of the following ways of pre-regions creation:

   - **stereopairs** to create pre-regions using overlap areas of two stereopair images;

     When using **stereopairs** frames the system fills in polygons attributes automatically as follows: *region_image_code* (with the left image code), *region_image_name* (with the left image name), region_image_code_2 (with the right image code), region_image_name_2 (with the right image name).

   - **images** to create pre-regions by near-nadir areas of each image.

     When using **images** frames the system fills in polygons attributes automatically as follows: *region_image_code*, *region_image_name*.

3. Select a way to create pre-regions using selected frames: Using **Voronoy diagram** or **version 4.x algorithms** earlier system's version algorithm (PHOTOMOD 4.x).

4. [optional] Set the **Remove slivers** slider (at **min** value the pre-regions are built considering narrow strips; at **max** value the pre-regions is created without narrow strips).

5. Click OK to create pre-regions. The system creates a new vector layer with pre-regions.

   To load, edit, save and close of pre-regions layer the system provides standard too for work with vector objects (see description in the *'Vector objects editing*' chapter of the Vectorization User Manual).

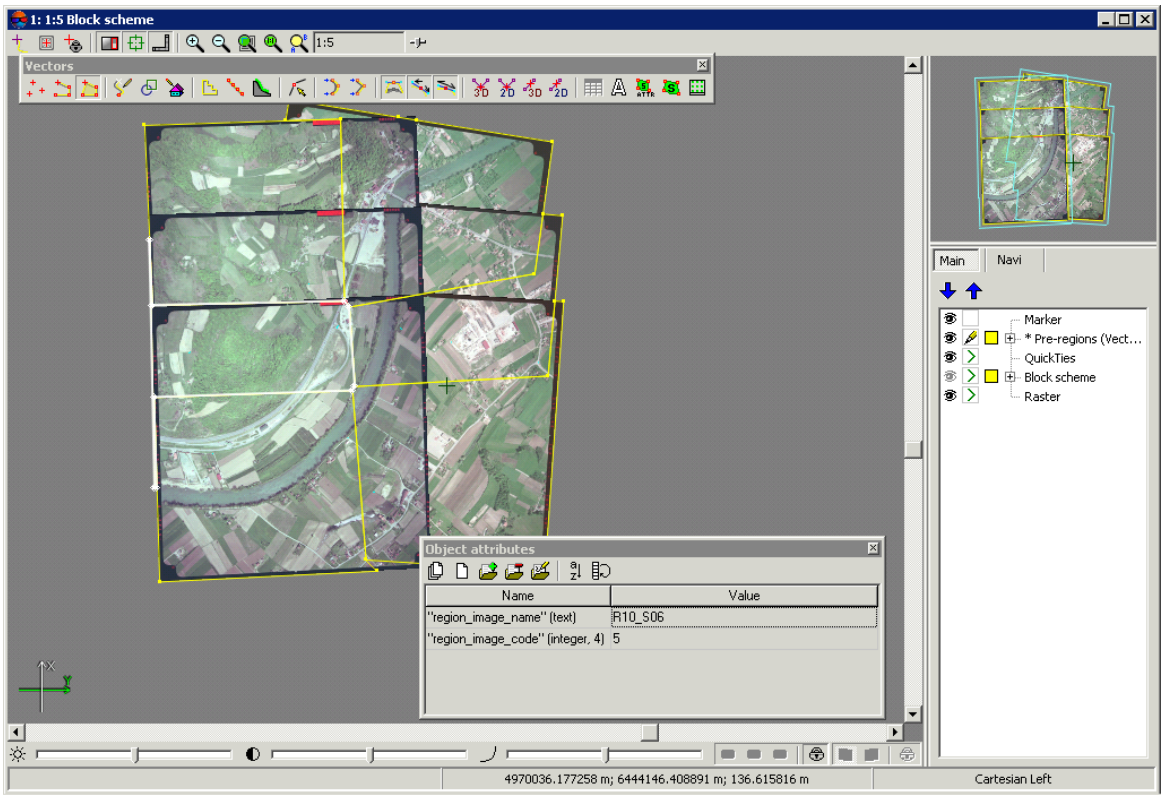


Fig. 26. Pre-regions created using images frames

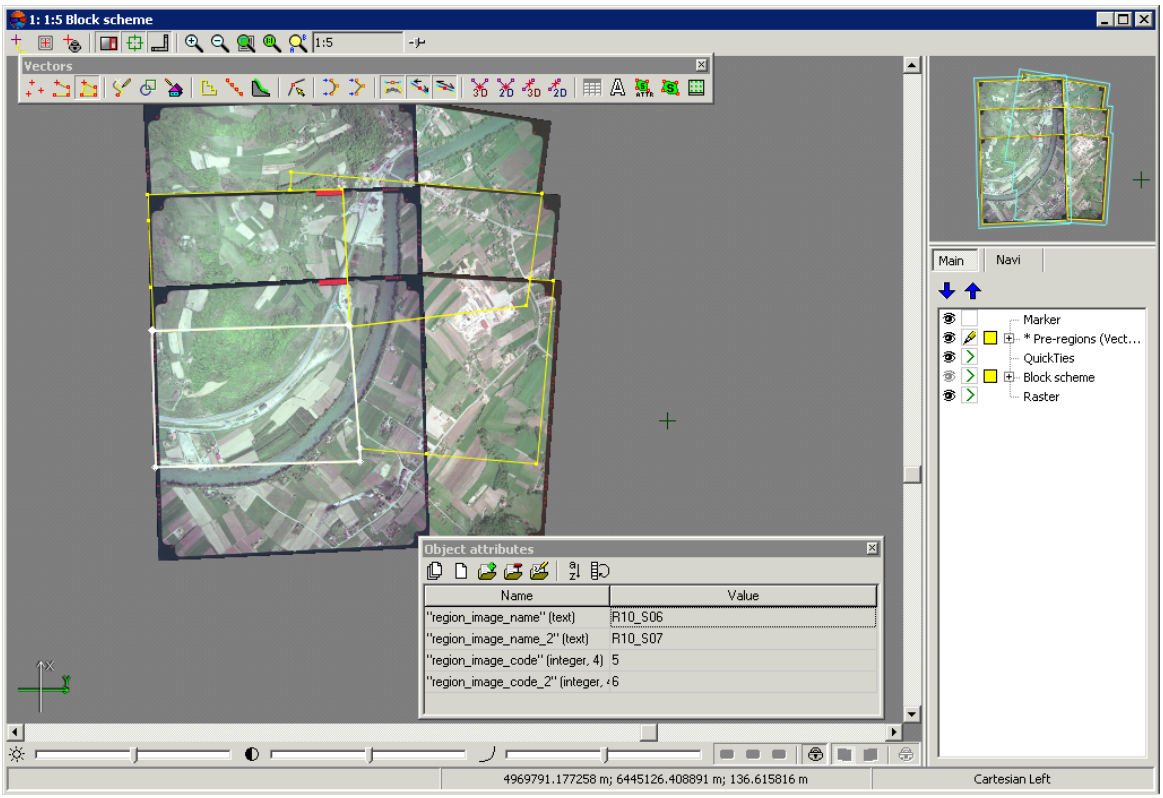


Fig. 27. Pre-regions created using stereopairs frames

5.3.2. Loading base layers for TIN creation

Prior to TIN creation it is necessary to define a set of base layers used to create a TIN.

The system performs TIN creation using layers, containing the following data:

- vector objects (points, polylines and polygons);
- breaklines – polylines/polygons, drawn along lengthy relief features (edges, watershed lines and valley-lines), as well as along roads;
- set of regular pickets – point vector objects, located on relief surface by regular grid or in arbitrary order;
- *triangulation points* – tie, check, GC points, acquired on step of aerial triangulation (see the 'Aerial triangulation' User Manual).

In order to load vector objects perform the following actions:

1. Choose **Vectors › Load** or click the button of the main toolbar. The **Open** window opens.

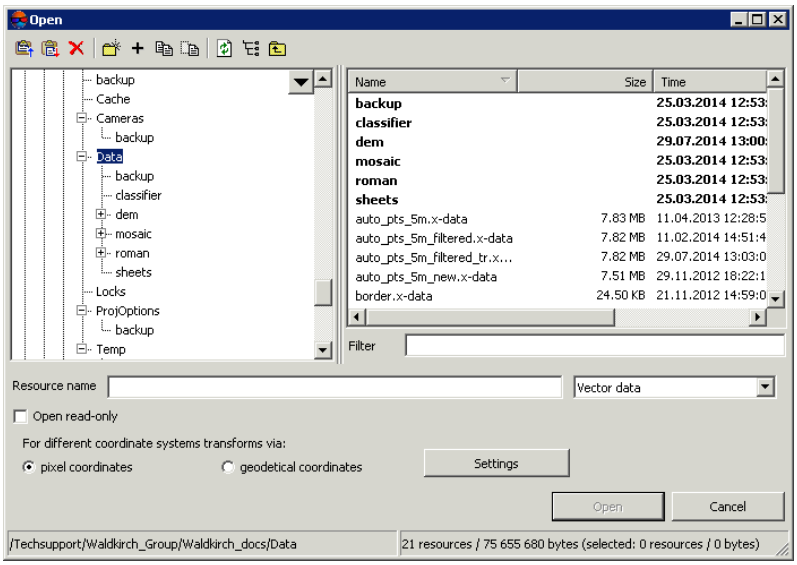


Fig. 28. Vector layer loading

2. Select one or multiple vector files in active profile resources with `x-data` extension.

3. [optional] To disable saving and rewriting of selected file, set the **Open read only** checkbox on.

4. [optional] Set method of objects coordinates recalculation **For different coordinate systems transforms via** – using pixel or geodetic coordinates (see details in the ’General system’s parameters’ User Manual).

5. Click the **Open** button. If one or multiple vector layers are already loaded to active layer, then during loading of new vector layer the window of loading type is opened.

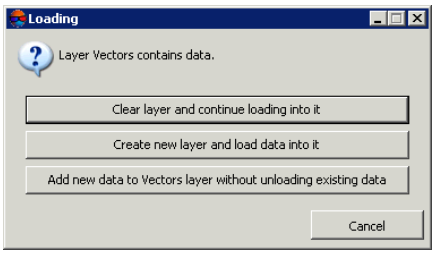

Loading
Layer Vectors contains data.
Clear layer and continue loading into it
Create new layer and load data into it
Add new data to Vectors layer without unloading existing data
Cancel

Fig. 29. Vector layer loading

The system provides the following possibilities of new layer loading to existing vector layer:

- **Clear layer and continue loading into it** – vector data of active layer is replaced by data from loading layer;

- **Create new layer and load data into it** – vector data is loaded to a new layer;

- **Add new data to Vectors layer without unloading existing data** – during loading vector data is added to objects of active vector layer.

If some point of vector objects layer cannot load, the system shows warning about failed loading of this points. In this case vector layer is loaded without coordinates of this point.

When it is necessary to load two or more vector files at once, the system suggests the following ways of loading:

- **Clear layer and load all files into it** – vector data of active layer is replaced by data from loading layers;
- **Load all into one new layer** – vector data is loaded to a single new vector layer;
- **Create separate layer for each file** – vector data of each file are loaded to separate layers;
- **Load all layers into Vectors without unloading existing data** – during loading vector data is added to objects of active vector layer.

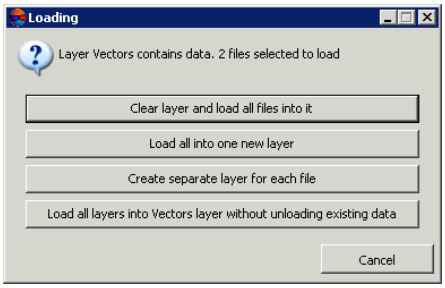

Loading
Layer Vectors contains data. 2 files selected to load
Clear layer and load all files into it
Load all into one new layer
Create separate layer for each file
Load all layers into Vectors layer without unloading existing data
Cancel

Fig. 30. Vector layer loading

Corresponding *Vector* layers are displayed in *Manager*.

If vector data layer is saved to different coordinate system or in a project with different adjustment results, the system recalculates coordinates automatically and during loading produces information message, that coordinates were recalculated.

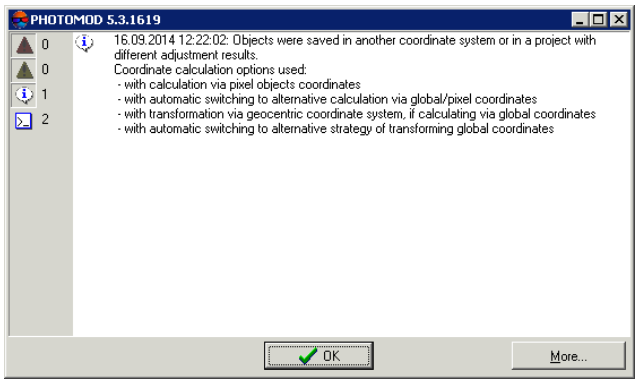

PHOTOMOD 5.3.1619
0
0
1
2
16.09.2014 12:22:02: Objects were saved in another coordinate system or in a project with different adjustment results.
Coordinate calculation options used:
- with calculation via pixel objects coordinates
- with automatic switching to alternative calculation via global/pixel coordinates
- with transformation via geocentric coordinate system, if calculating via global coordinates
- with automatic switching to alternative strategy of transforming global coordinates
OK
More...

Fig. 31. Coordinates recalculating during vector layer loading

TIN is rebuilt automatically during editing of base layers containing vector objects.

TIN layer is closed during closing of one of base layers.

To use a layer containing triangulation points, perform the following actions:

1. Choose **Vectors › Create layer from triangulation points**. The **Load triangulation points** window opens.

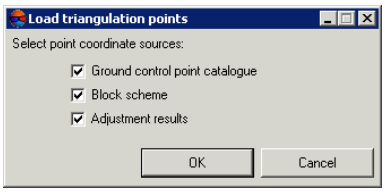


Fig. 32. Load triangulation points to a project

2. [optional] To export triangulation points from GCP list (see the) the **Ground control point catalogue** checkbox is set by default.

3. [optional] To export triangulation points from the *Block scheme* layer in 2D-window, the **Block scheme** checkbox is set by default.

4. [optional] To export adjusted points coordinates from GCP catalogue the **Adjustment results** checkbox is set by default.

5. Click OK. The system displays information window containing information about added or skipped points.

6. Click OK. The *Triangulation points (Vectors)* layer is created. Vector point objects (triangulation points) are displayed on block scheme in 2D-window.

Tie points are shown in 2D-window by yellow color; GCP – by red.

There is a possibility to save triangulation points layer to file of `*.x-data` format to active profile resources.

5.3.3. TIN borders creation

The system provides possibility of TIN borders creation manually or automatically.

To create TIN borders manually, create the border polygons on a separate layer and choose this layer in the list **Use polygons from layer** of the **Create TIN** window.

Perform the following actions to create TIN borders in automatic mode:

1. Choose **TIN › Compute border...**. The **Compute TIN border** window opens.

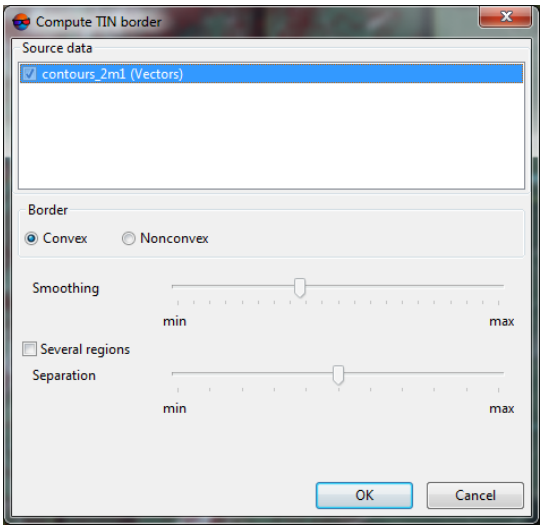


Fig. 33. The 'Compute TIN border' window

2. [optional] In the **Source data** section the checkboxes near all opened layers are set on by default, that are used for TIN borders creation. Set unused checkboxes off, if some layers opened are not used for TIN border creation.

3. [optional] Choose the type of TIN border:

   - **Convex**;
   - **Nonconvex**. To select smoothing level of nonconvex border move the **Smoothing** slider.

4. [optional] In order to split TIN border into areas depending on source data, set the **Several regions** checkbox on and move the **Separation** slider to define maximal distance between initial vector objects, used to create TIN border.

5. Click OK. The system starts to create a TIN border in a new *TIN border* layer.

5.3.4. Building TIN

Perform the following actions to build TIN:

1. Choose the **TIN › Create** (**Ctrl+N, T**) or click the button of the **DEM** additional toolbar. The **Create TIN** window opens.

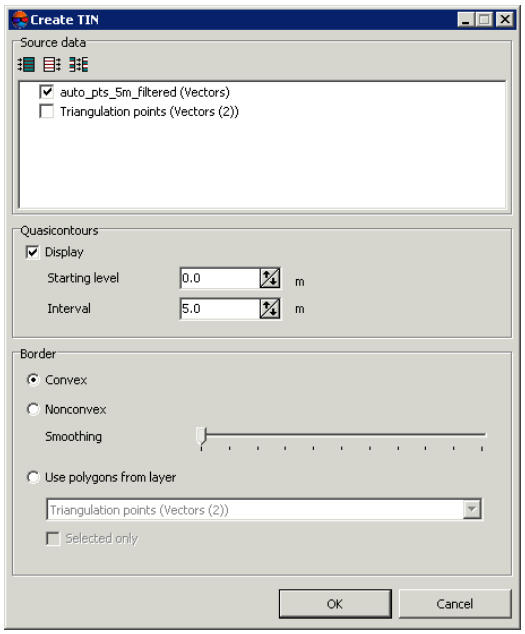


Fig. 34. The 'Create TIN' window

2. [optional] By default in the **Source data** section set on checkboxes for all opened layers to use them in TIN creation. Set the checkboxes off, if some layers opened are not used for TIN creation.

To select all available layers click the button, to unselect all layers – click the button. To invert selecting layers it is used the button.

3. [optional] In the **Quasi-contours** section the **Display** checkbox on and in the **Starting level** filed specify minimal Z level (Zmin), from which to start quasi-contours creation.

4. [optional] Specify the **Step** of quasi-contours creation in meters.

5. In the **Border** section set on the following:

   - **Convex** – the convex border of TIN is created;

     It is recommended to create convex border, when some part of vector objects does not cover all area of TIN creation (for example, when there are lakes and rivers on large scale images).

   - **Nonconvex** – the nonconvex border of TIN is created;

     It is recommended to place the **Smoothing** slider in the middle and smoothly move it to the left to get the best results.

   - **Use polygons from layer** – to create a TIN with convex border in limits of area of polygon(s) selected in the layers list. Select from the list a layer containing polygons, that will be used as a border. In order to define a border of TIN creation area, just from selected polygons, set the **Selected only** checkbox on.

6. Click OK. The TIN is created in the new *TIN* layer.

5.4. TIN display

The system provides possibility to setup TIN display in 2D-window.

In order to make settings choose menu item **Service › Settings** or click the button of the main toolbar. The **Settings** window opens. It consists of a set of tabs used to setup various parameters.

The **TIN** tab is used to setup parameters of TIN display

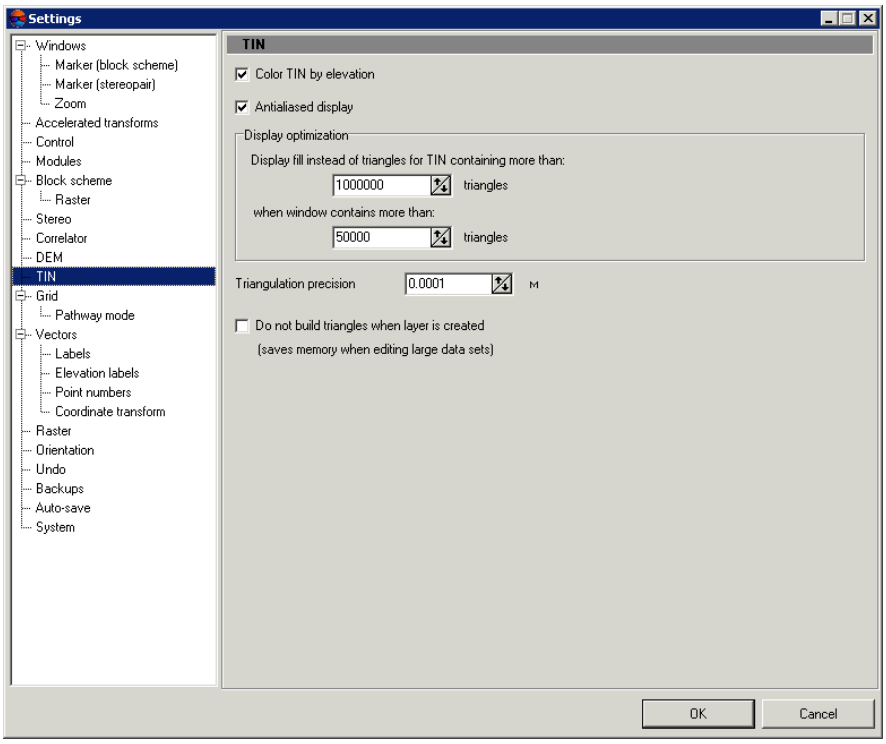


Fig. 35. Settings of TIN display

The **Color TIN by elevation** checkbox is set in the system be default, as a result TIN is displayed as grid of triangles colored by height. Clear the **Color TIN by elevation** checkbox to display all TIN triangles by the same color, defined in the **Manager** panel for TIN layer.

The **Display optimization** section contains the following parameters used to improve performance and to display a TIN as solid fill:

- **Display fill instead of triangles for TIN containing more than** specified number of triangles;
- Display solid fill for TIN **when window contains more than** specified number of triangles.

The **Antialiased display** checkbox is used to smooth TIN and quasi-contours for improving visual perception.

When TIN is displayed as solid fill quasi-contours are not displayed.

The **Triangulation precision** parameter is intended for filtering of errors of TIN creation using vector objects, which contain closely-spaced nodes. A value in the **Triangulation precision** field is used to specify minimal distance between points, at which the points are considered as different nodes, not the same one.

To save computer memory when editing large datasets set the **Do not build triangles when layer is created**.

If it is necessary to display TIN as triangles when the **Do not build triangles when layer is created** checkbox is on, in the **Display optimization** section input the **Display fill instead of triangles for TIN containing more than** parameter with value that exceeds the previous one.

The system provides a possibility to display TIN as three-dimension relief model, to do this select **Window › 3D window**.

5.5. TIN loading

In order to load a TIN perform the following actions:

1. Select **TIN › Load** (**Ctrl+O, T**) or click the button of the **DEM** additional toolbar. The **Open** window opens.

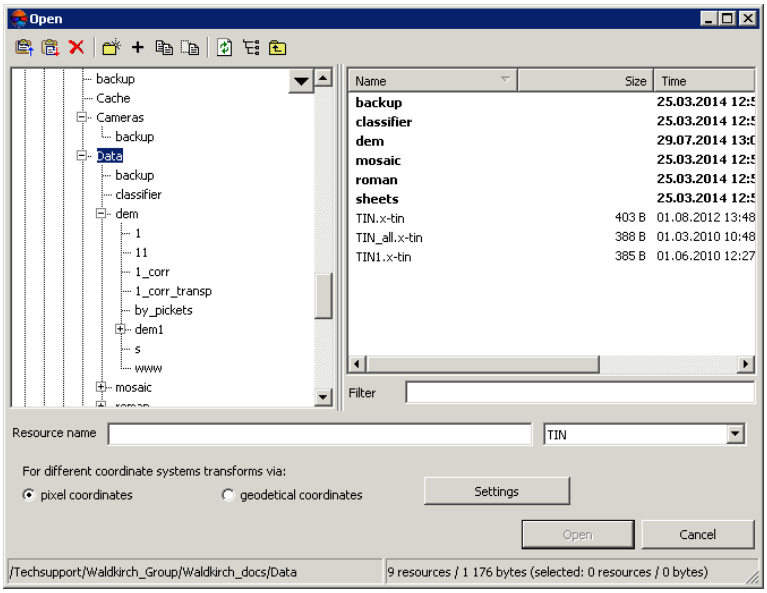


Fig. 36. TIN loading

2. Select one or multiple TIN files in active profile resources with `x-tin` extension.

3. [optional] Set method of objects coordinates recalculation **For different coordinate systems transforms via** – using pixel or geodetic coordinates (see details in the 'General system's parameters' User Manual).

4. Click the **Open** button. After that the TIN layer is loaded along with base vector layers. If one or multiple TIN containing layers are already loaded to active layer, then during loading of a new layer the window of loading type is opened.

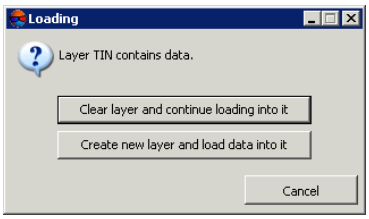


Fig. 37. TIN layer loading

The system provides the following possibilities of new layer loading to existing TIN layer:

- **Clear layer and continue loading into it** – vector data of active layer is replaced by data from loading layer;
- **Create new layer and load data into it** – vector data is loaded to a new layer.

The system provides a possibility of quick access to recently loaded TIN files. To do this use the **TIN › Recent** menu item.

5.6. TIN saving

To save and rewrite active TIN layer select **TIN › Save** or click the TIN layer name by mouse right button in the *Manager* and choose the **Save** in right-click menu.

Layer containing TIN is saved in the file with `x-tin` extension. The file contains paths to base layers used to create TIN, and a step of quasi-contours in meters.

To save active TIN layer with new name select **TIN › Save as...** or click the TIN layer name by mouse right button in the *Manager* and choose the **Save as...** in right-click menu, specify file name and destination in active profile resources.

During saving a layer containing TIN, base layers used for TIN creation are saved automatically.

During closing of one of base layers, the system suggests to save a layer containing TIN to active profile resources, after that the TIN layer is closed automatically.

5.7. TIN info

5.7.1. Brief TIN info

To get brief information about TIN layer, click the TIN layer name by mouse right button in the *Manager* and choose the **Information** in right-click menu. After that the system displays information window which includes the following information about the layer:

- layer name;

- full path to layer file;
- presence of data and changes in the layer;
- possibility to change the layer;
- coordinates of TIN border nodes, including Z-coordinate;
- number of nodes in base layers;
- layer coordinate system.

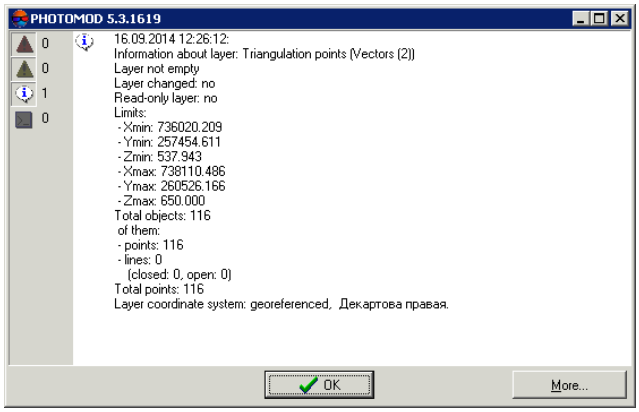


Fig. 38. TIN layer info

5.7.2. General TIN info

In order to obtain detailed information about TIN layer, choose **TIN › Statistics**. The **TIN – statistics** window opens.

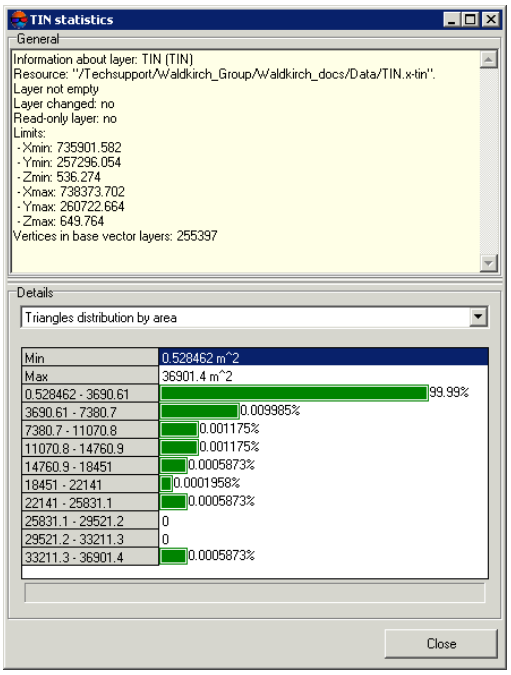


Fig. 39. The 'TIN – statistics' window

The **General** section includes brief information about TIN layer.

The **Details** section contains information about distribution range of the following data on a layer:

It is recommended to use this information to search for blunders on TIN layer.

- **Triangles distribution on area** (in square meters);
- **Nodes distribution along Z axis** (in meters);
- **Triangles distribution by min. angle** (in degrees);
- **Triangles distribution by max. angle** (in degrees).

Below is a table of the data distribution, which contains the following lines:

- **Min** and **Max** – minimal and maximal data range;
- (for example, `28588.2 - 31764.6`) – data distribution in percentage.

5.7.3. TIN area info

The system provides possibility to calculate TIN surface area.

To calculate TIN surface area within polygon, select the polygon on base layer or create a new polygon and select **TIN › Polygon area on surface**. After that the system opens information window with information about value of TIN area within selected polygon.

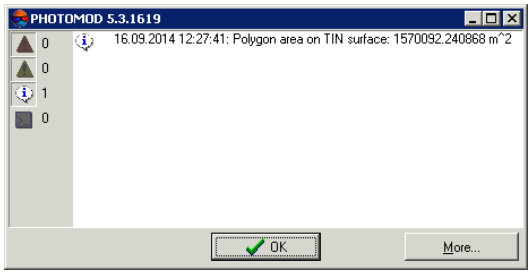


Fig. 40. The area of a polygon on TIN surface

In order to calculate area of TIN projection on a plane and area of TIN 3D surface, select **TIN › Compute area**. After that the system displays information window with values of TIN areas:

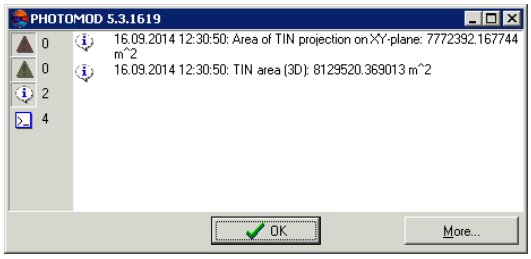


Fig. 41. TIN surface area

5.8. TIN re-building

After editing of base layer objects (see the 'Vectorization' User Manual) the system allows to perform TIN re-build operation.

TIN layer is not editable. To change TIN layer it is necessary to edit base layers used for TIN creation.

In order to re-build TIN considering vector objects of new vector layer, select **Vectors › Load**, select a file and click the **Open** button. The **Loading** window opens.

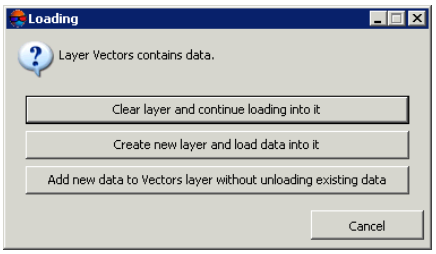

Loading
Layer Vectors contains data.
Clear layer and continue loading into it
Create new layer and load data into it
Add new data to Vectors layer without unloading existing data
Cancel

Fig. 42. Vector layer loading

The system provides the following possibilities of new layer loading to existing vector layer:

- **Clear layer and continue loading into it** or **Add new data to Vectors layer without unloading existing data** – allows to load vector objects to a base layer. After that the TIN is rebuilt automatically;
- **Create new layer and load data into it** – allows to load vector objects to a new layer. Create a TIN using new layer as a base one.

To re-build TIN considering changes made in vector objects located in base layers, perform the following actions:

1. Choose **TIN › Rebuild** or click the button of the **DEM** additional toolbar. The **TIN border smoothing** window opens.

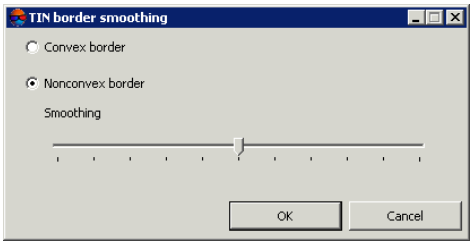


Fig. 43. TIN border smoothing

2. Set the following parameters:
   - **Convex border** – a border is created with connecting of outer boundary pickets of selected layer in such a way that TIN has the most smooth border;

     It is recommended to create convex border, when some part of vector objects does not cover all area of TIN creation (for example, when there are lakes and rivers on large scale images).

   - **Nonconvex border** – only closest border pickets of a layer are connected during border creation.

     The **Smoothing** slider position allows to specify distance between border pickets, and if the distance between pickets is less than the specified, the border will be passed through them. In the leftmost position of the slider pickets are connected sequentially, and in the rightmost one the border looks like convex.

     It is recommended to place the **Smoothing** slider in the middle and smoothly move it to the left to get the best results.

3. Click OK to perform TIN re-build according to changed data.

5.9. Insert objects into TIN

The system provides possibility to insert objects into existed TIN layer.

It is used, e.g. to display walls of buildings vertically in the TIN layer.

To insert vector objects into TIN layer is used the **TIN › Insert objects** menu item.

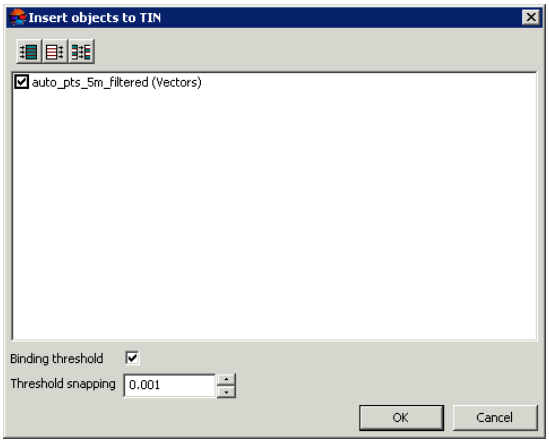


Fig. 44. Insert objects to TIN

To choose vector file are used the following buttons:

- – allows to select all files;
- – allows to unselect all files;
- – allows to invert files selection.

New layer creates when process complete. If the **Binding threshold** checkbox is set on, the new layer with inserted objects and TIN is created. Otherwise, only vector objects are created in new layer.

Threshold snapping allows to set the value in meters. If the value less than threshold, objects are not insert.

5.10. Accuracy control of TIN creation

5.10.1. Check against adjustment points

If triangulation points layer was not used during TIN creating, the system provides possibility to perform accuracy control of TIN creation by triangulation points, acquired on aerial triangulation step. To do this use the **TIN › Check against adjustment points** menu item.

The **Accuracy control** window includes a table of all points (GCP/check/tie), acquired on on aerial triangulation step (see the 'Aerial triangulation' User Manual).

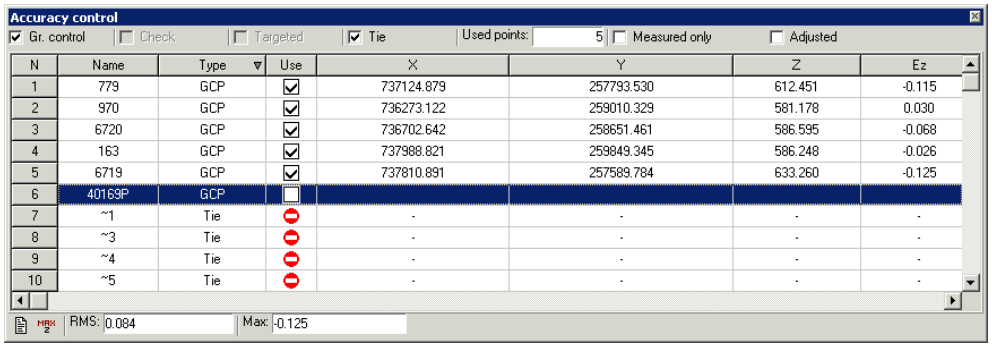


Fig. 45. Accuracy control using triangulation points

The top toolbar of the window is used to setup of triangulation points display in the list: **Gr.control**, **Check**, **Tie**, **Targeted**, **Measured only** (triangulation points with Z coordinate calculated during TIN creation), **Adjusted**.

The **Used points** field is used to display number of points used for calculation of TIN deviation by Z-coordinate (in the **Use** column the ☑ is set).

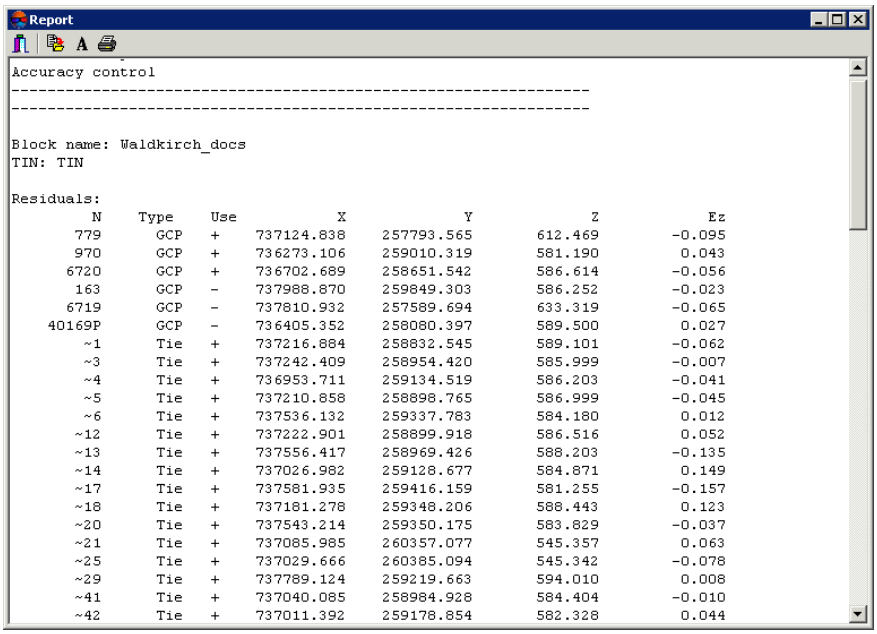
The table of triangulation points contains the following columns:

- **Name** – point number;
- **Type** – point type: GCP, check or tie point;
- **Use** – Data about point use during calculating of TIN deviation by Z-coordinate, or about adjustment:
  - ☑ – the triangulation point was used;
  - ☐ – the triangulation point was not used;
  - ⛔ – the triangulation point cannot be used, since it was not used for adjustment.
- **X**, **Y**, **Z** – adjusted coordinates of triangulation points;
- **Ez** – value of TIN deviation from the triangulation point by Z-coordinate.

Status panel is used to display values of root mean square error (**RMS**) and a value of maximal error of TIN deviation by Z-coordinate (**Max**).

The MAX button allows to display in the list a point with maximal error of TIN deviation by Z-coordinate.

The button allows to display detailed report on accuracy control.



N	Type	Use	X	Y	Z	Ez
779	GCP	+	737124.838	257793.565	612.469	-0.095
970	GCP	+	736273.106	259010.319	581.190	0.043
6720	GCP	+	736702.689	258651.542	586.614	-0.056
163	GCP	-	737988.870	259849.303	586.252	-0.023
6719	GCP	-	737810.932	257589.694	633.319	-0.065
40169P	GCP	-	736405.352	258080.397	589.500	0.027
~1	Tie	+	737216.884	258832.545	589.101	-0.062
~3	Tie	+	737242.409	258954.420	585.999	-0.007
~4	Tie	+	736953.711	259134.519	586.203	-0.041
~5	Tie	+	737210.858	258898.765	586.999	-0.045
~6	Tie	+	737536.132	259337.783	584.180	0.012
~12	Tie	+	737222.901	258899.918	586.516	0.052
~13	Tie	+	737556.417	258969.426	588.203	-0.135
~14	Tie	+	737026.982	259128.677	584.871	0.149
~17	Tie	+	737581.935	259416.159	581.255	-0.157
~18	Tie	+	737181.278	259348.206	588.443	0.123
~20	Tie	+	737543.214	259350.175	583.829	-0.037
~21	Tie	+	737085.985	260357.077	545.357	0.063
~25	Tie	+	737029.666	260385.094	545.342	-0.078
~29	Tie	+	737789.124	259219.663	594.010	0.008
~41	Tie	+	737040.085	258984.928	584.404	-0.010
~42	Tie	+	737011.392	259178.854	582.328	0.044

Fig. 46. Report on accuracy control using triangulation points

The top panel of the **Report** window contains the following buttons:

- – allows to close report window;
- – allows to save a report to file system of *Windows*;
- – allows to change font, style, size, and color of report text;
- – allows to print a report.

5.10.2. Check TIN against vector objects

The system allows to control TIN creation accuracy by vector objects, which were not used during TIN creation Control operation provides values of TIN deviation from vector objects vertices by Z-coordinate.

To control TIN creation against vector objects perform the following actions:

1. Select **TIN › Check TIN against vector objects**. The **Parameters of check TIN by vectors** window opens.

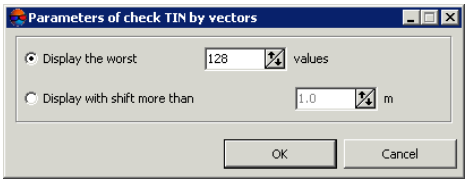


Fig. 47. Parameters of check TIN by vectors

2. Select display parameters of points list:

   - **Display the worst..values** – allows to display a list of vector objects vertices with worst deviations from TIN by Z-coordinate;

   - **Display the the shift more than** – allows to display a list of vector objects vertices with deviation from TIN by Z-coordinate, that exceeds specified.

3. Click OK. The system starts operation of comparing of TIN deviation values from vector objects vertices by Z-coordinate. After that the **Check TIN using vector objects** window opens.

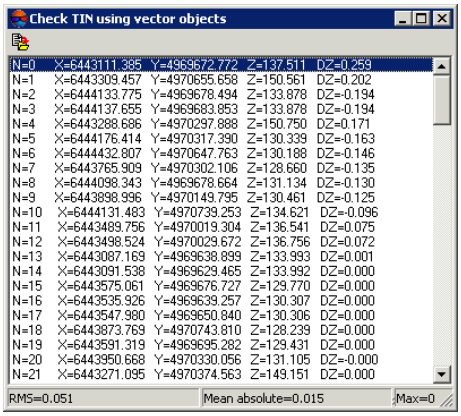


Fig. 48. Results of control check of TIN against vector objects

In order to export points list to file with `csv` extension, click the button.

4. Select a point in the list by mouse click. After that marker is placed to the crossing point and is displayed in 2D-window.

5.10.3. Verifying of TIN topology

The system provides possibility to verify TIN topology.

The *topology* concept is one of the key terms used during quality assessment of created maps. Throughout this documentation, the *topology* is referred to as set of functions and operations, that define mutual location of vector objects and are used to create topologically correct vector data.

Topology (from Greek *topos* - place) – a branch of mathematics that studies the phenomenon of continuity in the most general form, in particular, the properties of space, which remain unchanged under continuous deformations, for example, the connectivity.

Check of TIN topology means searching for closely placed triangles nodes of TIN and their correction (correction of their corresponding points or vertexes of polylines/polygons of base vector layers). Topology correction includes deleting or moving of closest TIN nodes.

In order to start TIN topology check, perform the following actions:

1. Select **TIN › Verify topology...**. The **Parameters** window opens.

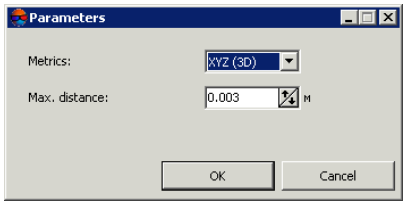


Fig. 49. Parameters of topology check

2. In the **Metrics** list specify whether 2D or 3D space is used to search for topology errors.

3. In the **Max. distance** field define a distance between nodes, below which the nodes are considered closest.

4. Click OK. The system starts operation of search for closest TIN nodes. When the search is completed the system opens the **Errors correction** window.

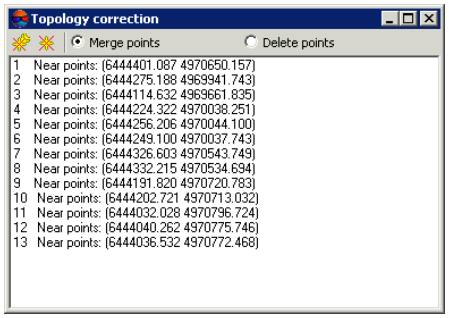


Fig. 50. Correction of topology errors

5. Select a point in the list by mouse click. After that marker is placed to the point and is displayed in 2D-window.

6. Select a way of topology correction: **Combine points** or **Delete points**.

7. To correct closest TIN nodes perform one of the following actions:

   - 'link' nodes using snapping (see the 'Vectorization' User Manual);
   - to correct topology for all points in the list click the button;
   - to correct topology only for selected points in the list click the button.

After correction of TIN closest nodes topology the erroneous nodes are combined into a single node with average coordinates value, or are deleted.

5.11. Export TIN

5.11.1. Export TIN to DXF

The system provides possibility to export of TIN layer to DXF format for its further use in third-party software.

In order to export TIN to DXF format perform the following actions:

1. Select **TIN › Export › DXF...**. The **Export to DXF** window opens.
2. Choose a folder to place a file to file system of *Windows* button.
3. Click the **Save** button. The **Export to DXF** window opens.

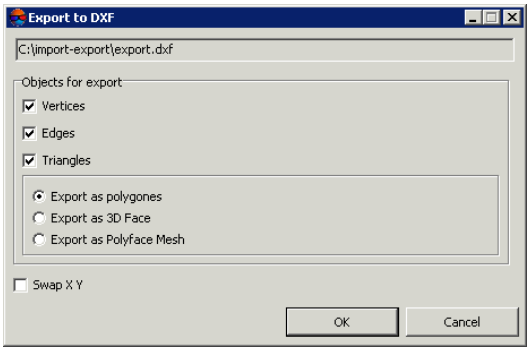


Fig. 51. Parameters of export to DXF format

4. [optional] In the **Objects for export** section to export vertices, edges and triangles of TIN the **Vertices**, **Edges**, **Triangles** checkboxes are set on by default. Clear the checkboxes to export just particular kind of objects.
5. Set the following parameters:
   - **Export as polygons** – allows to export objects as polygons;
   - **Export as 3D Face** – allows to export top face of an object;
   - **Export as Polyface Mesh** – allows to export object as polyhedral object.
6. [optional] In order to swap coordinate system (left/right) during TIN export, set the **Swap X Y** checkbox.

If the **Swap X Y** checkbox is off, the initial data will be imported in the right coordinate system. Otherwise, in the left coordinate system.

7. Click OK to complete the export operation.

5.11.2. Export TIN to CSV

The system provides possibility of TIN export to CSV format. CSV format is exchange text format with csv extension, which is supported by major applications in different industries. It is used as exchange format when special geospatial data formats are not applicable for some reason.

In order to export TIN to CSV format perform the following actions:

1. Select **TIN › Export › CSV...**. The **Export to CSV** window opens.
2. Choose a folder to place a file to file system of *Windows* button.
3. Click the **Save** button. The **Export to CSV** window opens.

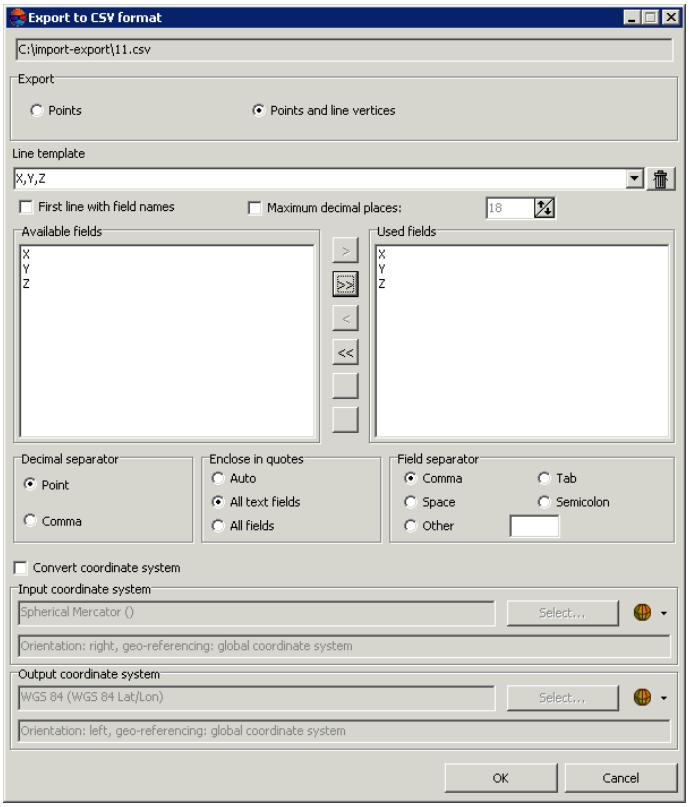


Fig. 52. Parameters of export to CSV format

4. In the **Export** section select what kind of objects it is necessary to export: **Points** or **Points and line vertices**.
5. The **Line template** field displays the list of fields, contained in each line of exported CSV file: To clear the field, click the **Clear the list** button.

6. [optional] In order to export a line from the **Line template** field, set the **First line with field names** checkbox on.

7. [optional] In order to change number of decimal places in coordinates to be exported, set the **Maximum decimal places:** checkbox on and input needed value.

8. In the **Available fields** list select required field name and click the **Add selected field** > button or click the **Add all fields** >> button, to transfer all field names. After that all or selected field names are moved to the **Used fields** list.

   To cancel field name selection click the < button or click the << button to cancel moving all field names. To move selected field to the list bottom, click the button, and to move selected field to the list top click the button.

9. In the **Decimal separator** section select point or comma to be used to separate coordinates.

10. [optional] In order to limit required parts of exported list of coordinates by quotes, in the **Enclose in quotes** section choose one of the following options:

    - **Auto** – allows to limit by quotes the fields, which are located in the file to be exported;
    - **All text fields** – allows to limit by quotes only the fields, which contain text information;
    - **All fields** – allows to limit by quotes each field, which is located in the exported file.

11. In the **Field separator** section choose, what is used to separate fields: comma, space, tab, semicolon or other delimiters.

    If the field separator is selected as a comma, it is not recommended to specify the decimal separator as a comma, since objects with incorrect coordinates will be created after import operation.

12. [optional] To define standard parameters set the **Transform reference system** checkbox on (see the chapter *Transform reference system of objects* in the 'Vectorization' User Manual);

13. Click OK to complete the export operation.

5.12. Filtering

5.12.1. Simplifying

The system allows to filter TIN triangles nodes using filtering of point vector objects in base layers, used for TIN creation.

TIN filtering is performed considering breaklines.

To perform pickets simplifying perform the following actions:

1. Create or load TIN.

2. Select **TIN › Simplify**. The **Simplify TIN** window opens.

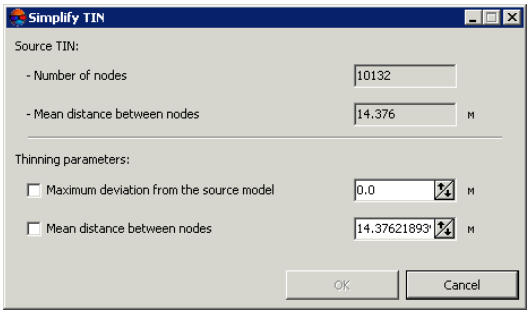


Fig. 53. Parameters of TIN simplifying

The window displays a statistics about source TIN: total **Number of nodes** in network and **Mean distance between nodes** (average length of TIN edges) in meters.

3. Set one or multiple checkboxes on and specify **Thinning parameters**:

   - **Maximum deviation from the source model** – maximum deviation between source and thinned models by Z in meters;

   - **Mean distance between nodes** – distance between nodes after thinning in meters.

4. Click OK. When the filtering operation is completed the system displays information message about number of deleted nodes.

Deleting of TIN nodes means removing of vector points from base layers. The filter works with all base layers as with a single editable layer.

To cancel thinning results the **Undo log** is used, as well as the and button of the main toolbar.

5.12.2. Peak filter

The system provides possibility to filter peaks. This feature is used to search for TIN nodes, which Z value differs considerably from adjacent nodes, so the system defines nodes with sharp local peaks. Peaks filtering allows to remove part of nodes, that form the most 'spiky' pyramids.

TIN filtering is performed considering breaklines.

Perform the following actions to filter peaks:

1. Create or load TIN.

2. Select **TIN › Peak filtering**. The **Peak filtering** window opens.

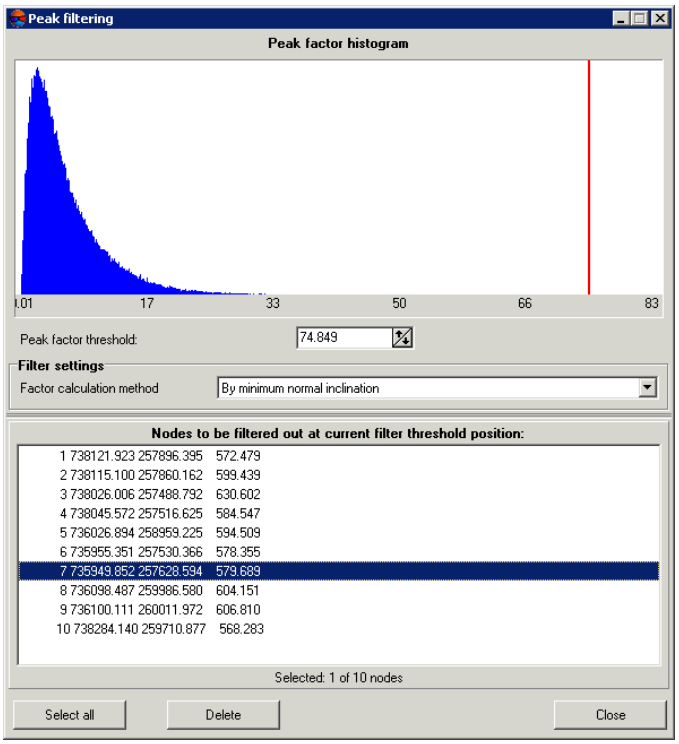


Fig. 54. Peaks filtering

The histogram shows the relative distribution of TIN nodes with different coefficients of peaks. A node with a value of peak factor, that equals 0, is assumed error-free. A node with maximal probability of error (in accordance with selected calculation method) have a peak factor that equals 100.

3. Specify **Peak factor threshold** or move using mouse button the red line on the diagram to filter nodes, that fall into areas with values exceeding specified (behind the red line).

4. In the **Filter settings** section select **Factor calculation method** in the list:

   - **By minimum normal inclination** – peak factor is proportional to the minimum factor by triangles surrounding the vertex, to the value of angle between the normal to the plane of the triangle and Z axis;
   - **By maximum normal inclination** – peak factor is proportional to the maximum factor by triangles surrounding the vertex, to the value of angle between the normal to the plane of the triangle and Z axis;
   - **By mean normal inclination** – peak factor is proportional to the mean factor by triangles surrounding the vertex, to the value of angle between the normal to the plane of the triangle and Z axis;
   - **By sum normal projection** – peak factor is inversely proportional to module projection of the vector sum of the normals to triangles surrounding the vertex on Z axis;
   - **By sum normal abs.value** – peak factor is inversely proportional to module projection of the vector sum of the normals to triangles surrounding the vertex.

5. Click the **Refresh** button. The bottom part of the window shows a list of nodes, that correspond to peaks with specified parameters.

6. Select a node in the list by mouse click. After that marker is placed to the selected node and is displayed in 2D-window.

   To select nodes group it is possible to use **Ctrl** or **Shift** keys. The **Select all** button allows to select all nodes found.

7. Click the **Delete** button to delete pickets from base layer selected in the list. After that the TIN is rebuilt automatically.

   Deleting of TIN nodes means removing of vector points from base layers. To cancel thinning results the **Undo log** is used, as well as the and button of the main toolbar.

5.12.3. TIN filtering by Z-range

The system provides possibility to edit TIN using Z-filter. During filtering by Z-range the system performs deleting of vector points, and polylines/polygons vertices (that correspond to TIN nodes), which Z-coordinate is out of the range.

TIN filtering is performed considering breaklines.

To perform filtering by Z-range of vector objects of base vector layers, used for TIN creation, perform the following actions:

1. Create or load TIN.

2. Select **TIN › Filter › Filter by Z-range**. The **Z-range filter** window opens.

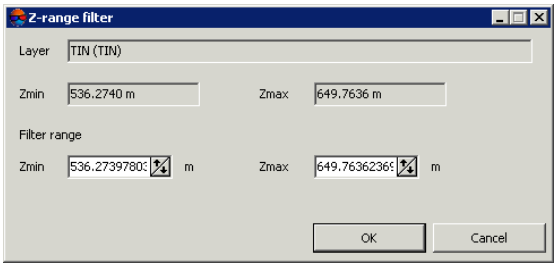


Fig. 55. Parameters of TIN filtering

The **Layer** field displays a name of loaded TIN layer, the **Zmin** and **Zmax** fields show terrain elevation difference in meters, calculated by all TIN nodes.

3. The **Filter range** section by default shows values of calculated terrain elevation difference. Specify maximal and minimal Z values in meters to consider for points filtering.

4. Click OK. Filtration removes all points and vertices of polylines/polygons, which Z-coordinate falls outside specified range.

Deleting of TIN nodes means removing of vector points from base layers. To cancel thinning results the **Undo log** is used, as well as the and button of the main toolbar.

5.12.4. Smooth TIN interpolation

The system provides possibility to interpolate TIN network for DEM smoothing for creating or improving contour lines, if contour lines layer was one of the base layers for TIN creation.

TIN filtering is performed considering breaklines.

Perform the following actions to perform TIN interpolation:

1. Create or load TIN.

To interpolate TIN within a grid it is necessary to create or load the grid **prior to** interpolation operation.

2. Select **TIN › Interpolate**. The **Smooth TIN interpolation** window opens.

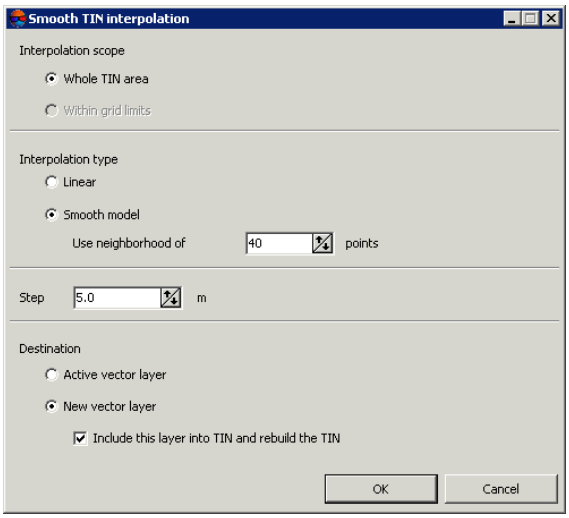


Fig. 56. Parameters of TIN interpolation

3. Specify the **Interpolation scope**:

   - **Whole TIN area**;
   - **Within grid limits** – the system uses parameters of regular grid with specified step.

4. Specify the **Interpolation type**:

   - **Linear** – TIN condensation using points adding on triangles surface;
   - **Smooth model** – using second-order surface for interpolation by specified number of points in the neighbourhood of the triangle, where a grid node is located.

     For smooth model specify a number of points used to perform interpolation in the **Use neighbourhood of..points** field.

     It is recommended to specify a value not exceeding 100 points in the **Use neighbourhood of..points** field.

5. Set the **Step** of interpolation in meters.

6. Select a destination of interpolation results save:

   - **Active vector layer** – the active vector layer will be rewritten;
   - **New vector layer** – interpolation results will be copied to a new vector layer.

The **Include this layer into TIN and rebuild the TIN** checkbox is set on by default, that allows to update TIN network automatically according to filtering results.

7. Click OK. After that the system performs interpolation of base vector layer and displays results depending on selected parameters.

6. Contour lines

6.1. Generation contour lines by TIN

The system provides possibility to create *contour lines* – vector lines, joining points with the same terrain elevation – by irregular spatial triangles network (TIN).

To create contour lines by TIN, perform the following actions:

1. Load TIN layer.

2. Choose **Vectors › Build contours › From TIN...** or use the **Ctrl+N, C** hotkeys. The **Create contours by TIN** window opens.

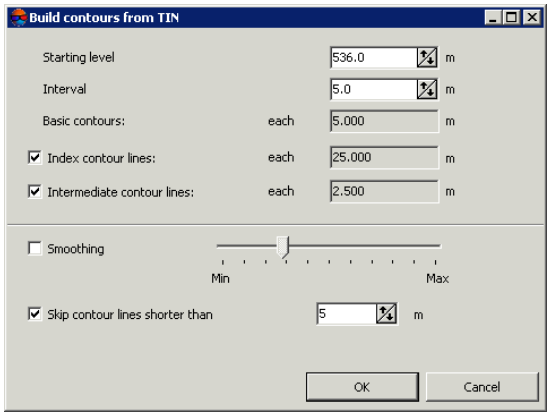


Fig. 57. Parameters of contour lines generation by TIN

3. [optional] Input minimal elevation level (Zmin), which will be zero-level for contours creation, to the **Starting level** field.

In order to obtain information about elevation difference (Zmin, Zmax), right click on the TIN layer in the *Manager* and select the **Information** item, or select **Edit › Active layer › Layer information**. The system displays the information window that contains information about the layer.

4. In the **Interval** field specify contours interval in meters. Interval of basic contours is calculated automatically and is displayed in the **Basic contours** field. If the **Intermediate contour lines** checkbox is off, interval of basic contours equals to contours interval.

5. [optional] To create index contour lines the **Index contour lines** checkbox is on by default. Set the checkbox off to cancel index contour lines creation.

   The interval of **index contour lines** is calculated considering the **Basic contours** parameter and equals to five intervals of basic contours.

6. [optional] In order to create intermediate contours, set the **Intermediate contour lines** checkbox on.

   The interval of **intermediate contour lines** is calculated considering the **Basic contours** parameter and equals to half of basic contours interval.

7. [optional] In order to create contours as smoothed curves, set the **Smoothing** checkbox on and specify a level of smoothing using the slider.

8. [optional] In order to cancel creation of short contours, set the **Skip contour lines shorter than** checkbox on and input minimal contour length in meters.

9. Click OK. After that the system starts to create contours in a new layer *Contours*.

To edit created contours select **Window › Contours classifier table** (see Section 6.5).

6.2. Contours creation in batch mode

The system provides possibility to create contour lines in batch mode. Batch mode allows to create multiple files of contour lines simultaneously and to save them to specified folder.

Contours creation in batch mode is possible in 64-bit system version only.

In order to build contours using TIN in batch mode perform the following actions:

1. Choose **Vectors › Build contours › In batch mode...**. The **Create contours in batch mode** window opens.

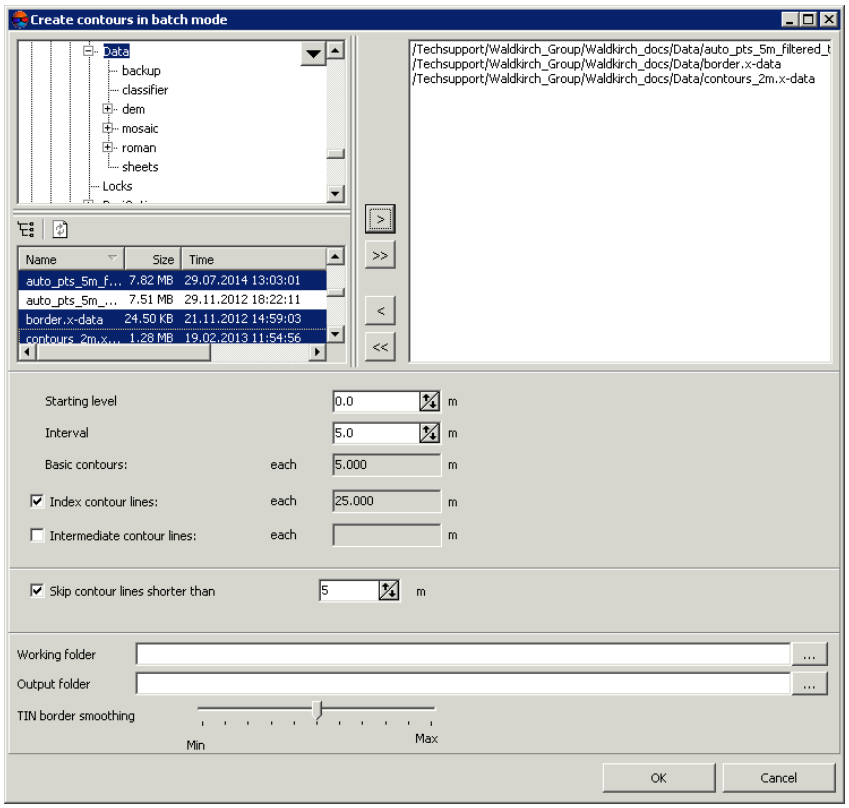


Fig. 58. Parameters of contours creation in batch mode

2. Choose a folder containing vector objects in resources tree.

   The button allows to display all available resources in nested files. The button allows to refresh part of the window with resources.

   The button allows to show a list with 10 recently selected resources.

3. Select file with contours in the list and click the > button to add a layer.

   The >> and << buttons allow to add to the list or remove from it all added files with vector objects, the < button allows to remove selected file from the list.

4. Repeat the 2 and 3 steps to add next files with vector objects.

5. Click the ... button in the **Working folder** field and select a folder in active profile resources.

6. Click the ... button in the **Output folder** filed and select a folder in active profile resources to save output contour lines.

7. Configure necessary parameters of contours creation like for **Generation contour lines by TIN**.

8. Click OK to start contour lines creation in batch mode.

To edit created contours select **Window › Contours classifier table**.

6.3. Generation contour lines by DEM

The system provides possibility to create contour lines by digital elevation model (DEM).

In order to build contours using DEM perform the following actions:

1. Load a DEM layer.

2. Choose **Vectors › Build contours › From DEM...**. The **Contours creation parameters** window opens.

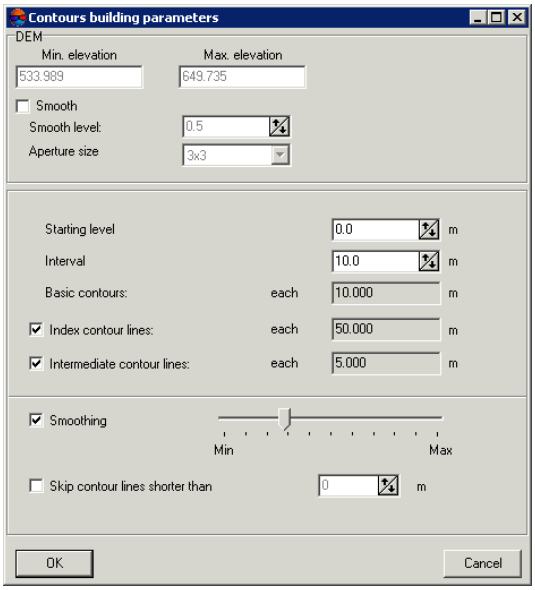


Fig. 59. Parameters of contour lines generation by DEM

3. [optional] In order to apply operation of DEM smoothing prior to contours creation, set the **Smooth** checkbox on in the **DEM** section. Specify smoothing level in the **Smooth level** field (maximal smoothing is applied at 1 value).

   During smoothing the system uses information about adjacent nodes in each DEM node. In order to specify number of adjacent nodes of DEM, select the **Aperture size** in the list.

   Min. elevation and **Max. elevation** – display information about elevation difference (Zmin, Zmax) in DEM.

4. [optional] Input minimal elevation level (Zmin), which will be zero-level for contours creation, to the **Starting level** field.

   In order to obtain information about about elevation difference (Zmin, Zmax), select **Edit › Active layer › Layer information** or click the DEM layer by mouse right button in the *Manager* and select **Information**. The system displays the information window that contains information about the layer.

5. In the **Interval** field specify contours interval in meters. Interval of basic contours is calculated automatically and is displayed in the **Basic contours** field. If the **Intermediate contour lines** checkbox is off, interval of basic contours equals to contours interval.

6. [optional] To create index contour lines the **Index contour lines** checkbox is on by default. Set the checkbox off to cancel index contour lines creation.

   The interval of **index contour lines** is calculated considering the **Basic contours** parameter and equals to five intervals of basic contours.

7. [optional] In order to create intermediate contours, set the **Intermediate contour lines** checkbox on.

   The interval of **intermediate contour lines** is calculated considering the **Basic contours** parameter and equals to half of basic contours interval.

8. [optional] In order to create contours as smoothed curves, set the **Smoothing** checkbox on and specify a level of smoothing using the slider.

9. [optional] In order to cancel creation of short contours, set the **Skip contour lines shorter than** checkbox on and input minimal contour length in meters.

10. Click OK. After that the system starts to create contours in a new layer *Contours*.

To edit created contours select **Window › Contours classifier table**.

6.4. Contours creation from smooth DTM

The system provides possibility to create contour lines as smoothed curve lines with specified accuracy. Using this method of contours creation allows to avoid contours intersection.

Source data for contours creation are vector objects obtained after stereo vectorization. If necessary the system allows to create a polygon and to use it then as a border of contours creation area.

In order to build contours from smooth DTM perform the following actions:

1. Load a layer containing vector objects.

2. Choose **Vectors › Build contours › Smooth model...**. The **Build contours from smooth DTM** window opens.

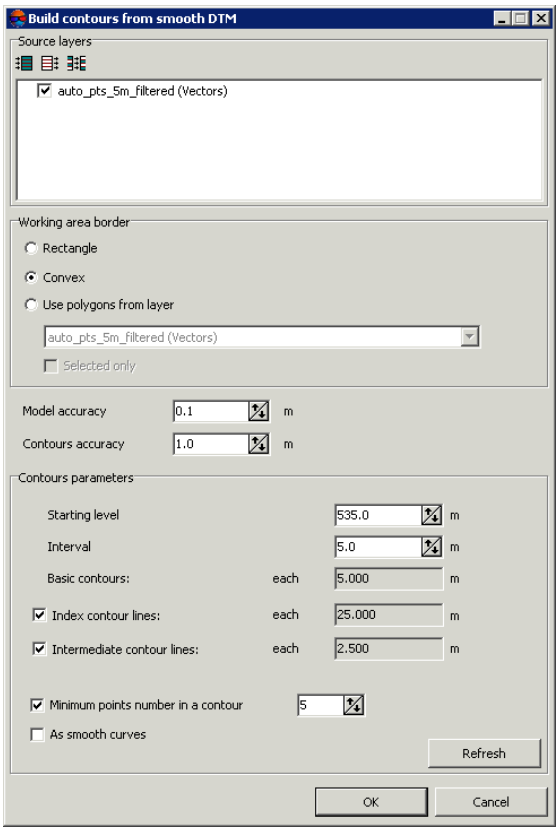


Fig. 60. Parameters of contours creation from smooth DTM

3. In the **Source layers** section set the checkboxes on next to opened layers used for contours creation.

4. In the **Working area border** layer specify border of contours creation area:

   - **Rectangle**;
   - **Convex**;
   - **Use polygons from layer** – select from the list a vector layer that contains polygons used as working area border.

   In order to define a border of contours creation area, just from selected polygons, set the **Selected only** checkbox on.

5. In the **Model accuracy** field input acceptable deviation of contours elevation values from source vector layer in meters.

   The **Model accuracy** parameter should be specified based on errors values obtained during creation of vector objects using correlator in stereomode.

6. In the **Contours accuracy** field input maximal deviation of contours from DEM by Z.

7. [optional] Input minimal elevation level (Zmin), which will be zero-level for contours creation, to the **Starting level** field.

   In order to obtain information about elevation difference (Zmin, Zmax), right click on the TIN layer in the *Manager* and select the **Information** item, or select **Edit › Active layer › Layer information**. The system displays the information window that contains information about the layer.

8. In the **Interval** field specify contours interval in meters. Interval of basic contours is calculated automatically and is displayed in the **Basic contours** field. If the **Intermediate contour lines** checkbox is off, interval of basic contours equals to contours interval.

9. [optional] To create index contour lines the **Index contour lines** checkbox is on by default. Set the checkbox off to cancel index contour lines creation.

   The interval of **index contour lines** is calculated considering the **Basic contours** parameter and equals to five intervals of basic contours.

10. [optional] In order to create intermediate contours, set the **Intermediate contour lines** checkbox on.

    The interval of **intermediate contour lines** is calculated considering the **Basic contours** parameter and equals to half of basic contours interval.

11. [optional] To create contours with number of nodes more then specified value, the **Minimal points number in a contour** checkbox is set on by default. Input a number of nodes to the input field to exclude contours with less number of nodes.

12. [optional] To create contours as arc segments (curves) set the **As smooth curves** checkbox on.

13. Click OK. After that the system starts to create contours in a new layer *Contours*.

6.5. Contour lines editing

To edit contours display parameters the system provides **Window › Contours classifier table** menu item. Display parameters are saved and applied during creation or loading of new contours layer.

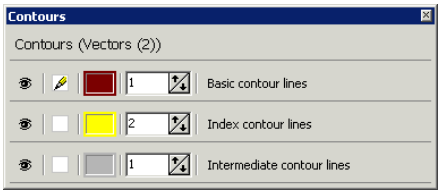


Fig. 61. Contours classifier

The **Contours** window allows to configure the following parameters:

- – allows to hide/show selected type of contours in 2D-window;
- – allows to select a type of contours to be edited;
- **Layer color** – allows to configure color to display selected contours type;

  To change contour color, double click the **Layer color** button. The **Color** window opens. Choose contour color and click OK.

- input field allows to specify width of selected contours type, that remains the same during scale change.

  To change a contour width, specify a value in this input field (this parameter remains the same during scale change).

6.6. Operations with contour lines

6.6.1. Saving contour lines

Contours are polylines that include straight line segments (broken lines) or arc segments (curves). Contours saving, loading and editing is performed in the same way as for any vector layer (see the Vectorizarion User Manual).

For contours editing the system provides tools of polylines editing, see description in the Vectorization User Manual.

To save and rewrite active contours layer select **Vectors › Save** or click the layer name by mouse right button in the *Manager* and choose the **Save** in right-click menu.

To save active layer with new name select **Vectors › Save as...** or click the layer name by mouse right button in the *Manager* and choose the **Save as...** in right-click menu, specify file name and destination in active profile resources.

To save *just* selected contours, select all contours that should be saved and select **Vectors › Save selected as...**. After that vector objects are still associated with classifier.

In order to return to vector layer state at the time of the last save, select **Vectors › Revert**.

The *Contours* layer is opened in the same way as a layer with usual vector objects, see Section 5.3.2.

6.6.2. Contours merging

The system provides possibility to merge created contours in automatic or manual mode.

In order to merge contours **manually**, perform the following actions:

1. Load or create a layer with contour lines.
2. Select **Vectors › Load** and open a file containing contour lines from adjacent stereopair. The **Loading** window opens.
3. Click the **Add new data to ... layer without unloading existing data** button to add contours from adjacent stereopair to a source layer.
4. Use tools intended for polylines editing (as well as snapping) for contours manual merging, see the Vectorization User Manual.

In order to start automatic contours merging, perform the following actions:

1. Select **Vectors › Contour operations › Merge contours...**. The **Parameters** window on the **Source contours** tab opens.

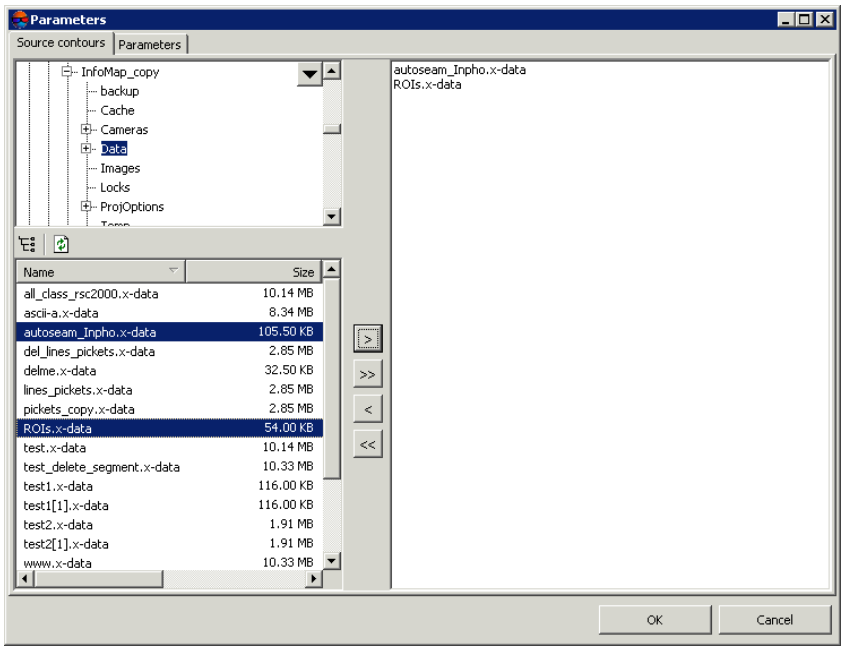


Fig. 62. Parameters of contours merging on the 'Source contours' tab

2. Choose a folder containing contours in resources tree.

The button allows to display all available resources in nested files. The button allows to refresh part of the window with resources.

The button allows to show a list with 10 recently selected resources.

3. Select file with contours in the list and click the > button to add a layer to be merged.

The >> and << buttons allow to add to the list or remove from it all added files with contours, the > button allows to remove selected file with contours from the list.

4. Repeat the 2 and 3 actions to add next files with contours.

It is necessary to choose for merging at least two files containing contour lines.

5. Click the **Parameters** tab.

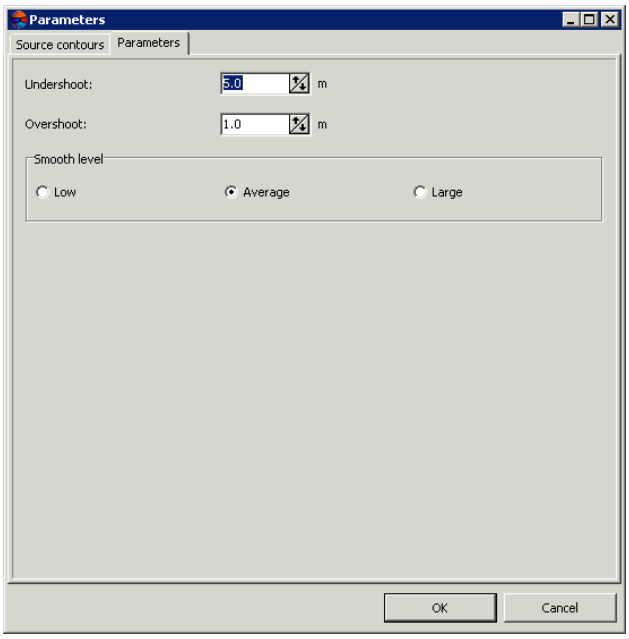


Fig. 63. Parameters of contours merging on the 'Parameters' tab

6. [optional] Input maximal distance between contours in meters, at which the system performs contours merging, to the **Undershoot** field.

7. [optional] To the **Overshoot** input acceptable part of contour line to be deleted in meters.

8. In the **Smooth level** section specify a smoothing level of smoothed contour from vertices of unsmoothed one: **Low**, **Average** or **Large**.

9. Click OK. New vector layer that contains merged contours is created in the *Manager*.

6.6.3. Check of contours merging quality

The system provides possibility to check contours for breaks that occur after manual or automatic contours merging.

To perform check of contours merging, select **Vectors › Contour operations › Check merging contours ...**. The **Parameters** window opens. In the **Check distance** input field specify maximal distance between contours in meters, when the system starts to search for contours breaks, and click OK.

If the contours breaks are detected the **Erroneous contours vertices** window opens.

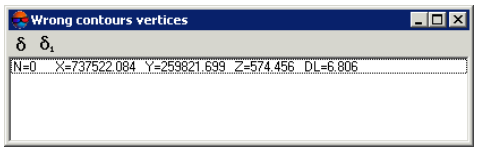


Fig. 64. Erroneous contours vertices

The window displays a list of contour lines breaks, where:

- `N` – is break serial number;
- `X,Y,Z` – coordinates of break's middle point;
- `DL` – distance between contours in meters.

Select a line in the list by mouse click. After that marker is placed to the break's middle point and is displayed in 2D-window.

To 'connect' contour break, select the contour in the list and click the δ_1 button.

To 'connect' all found contour breaks, click the δ button.

6.7. Accuracy control of contours creation

6.7.1. Control of contours crossing

The system allows to check created contours for crossing/self-crossing, that occur after operation of contours smoothing.

In order to check contours for crossings perform the following actions:

1. Load or create a layer with contour lines.
2. Select **Vectors › Contour operations › Check contours intersections...**. The **Parameters** window opens.

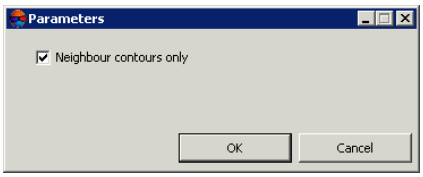


Fig. 65. Parameters of contours intersections control

3. [optional] To check intersections of adjacent contours only, the **Neighbour contours only** is set on by default. Set the checkbox off to check all contour lines for intersections.

Operation of created contours check for intersections may take a long time.

4. Click OK. The system start operation of created contours check for intersections. When the check is completed the system opens the **Intersection points** window.

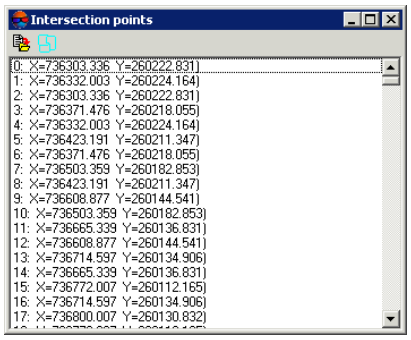


Fig. 66. Coordinates of intersection points of contour lines

In order to export a list of contours intersection points to file with `csv` extension, click the button.

To create a new layer that contains intersection points as pickets, click the button.

5. Select a point in the list by mouse click. After that marker is placed to the crossing point and is displayed in 2D-window.

6. Use tools of polylines editing to edit contours and correction of intersection\self-intersection errors (see description in the Vectorization User Manual).

6.7.2. Check contours by points

The system provides opportunity to perform quality control of contours creation using regular pickets, if they were not used for contours creation.

To start the check operation it is necessary that the *Contours* layer and the *Vectors* layer (containing pickets) are loaded and displayed in the *Manager*.

In order to start contours check operation by pickets, perform the following actions:

1. Load or create a layer with pickets.

2. Make active the layer with contour lines.

3. Select **Vectors › Contour operations › Check contours by points**. The **Select points layer** window opens.

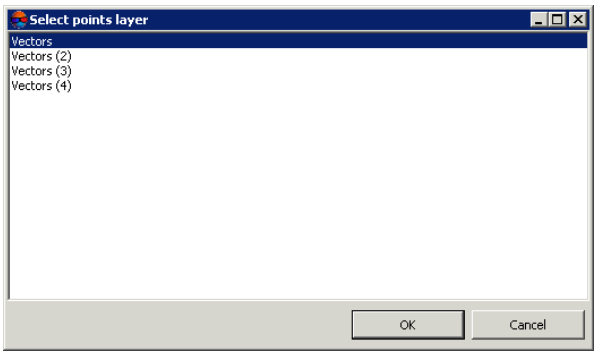


Fig. 67. Selection of pickets layer

4. Select a layer containing pickets.

5. Click OK. The **Parameters** window opens.

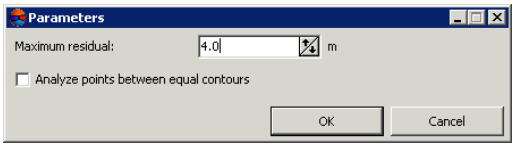


Fig. 68. Parameters of contours check using pickets

6. In the **Maximum residual** field input a value of maximal deviation of contours from a value of pickets elevation by Z (in meters).

7. [optional] In order to include to check operation pickets located on contours with the same elevation (for example, `N=27 (Ext=597.0, 597.0)` and `N=28 (Ext=597.0, 597.0)`), set the **Analyse points between equal contours** checkbox on.

8. Click OK. The **Check contours by points** window opens.

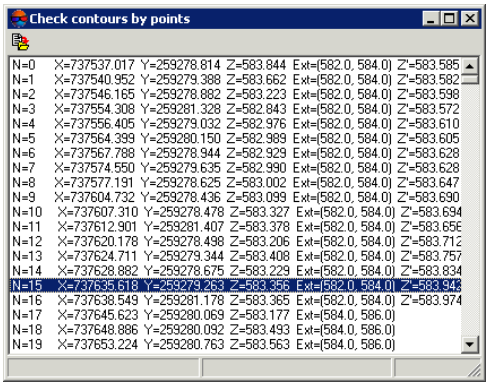


Fig. 69. Points coordinates during control of contours using pickets

The window displays a list, where:

- `N` – is picket serial number;
- `X,Y,Z` – picket coordinates;
- `Ext` – distance between contours in XY plane;
- `Z'`– elevation difference between contours and pickets, that exceeds maximal deviation.

In order to export the list to file with `csv` extension, click the button.

9. Select a picket in the list by mouse click. After that marker is placed to the selected picket and is displayed in 2D-window.
10. Use tools of polylines editing to edit contours (see description in the Vectorization User Manual).

7. Digital Elevation Model

7.1. The ’DEM’ menu

The **DEM** menu contains standard menu items used to load and save DEM layers, as well as menu items used to perform various operations on DEM creation, accuracy control, filtering and editing.

Table 3. Brief description of the DEM menu

Menu items	Function
Build DEM...	contains menu items used to create DEM with help of various source data
Rebuild by TIN	allows to recreate DEM after changing of TIN base layer
Restore	is used to restoring of DEM consistency, which may be disrupted by applying various operations or DEM filtering
Buildings and vegetation filter	is used to Buildings and vegetation filtering in DEM
Median filter	allows to use median filtering of DEM cells
Smooth filter...	allows to use smoothing filter for DEM values
Fill null cells	contains menu items to restore blank cells of DEM using various methods
Convert cells to null	allows to convert DEM cells with specified elevation to null ones
Set elevation in selected polygons...	allows to set the same elevation of DEM cells inside and outside of selected polygons
Interpolate elevation in selected polygons...	allows to interpolate a value of DEM cells inside and outside of selected polygons
Cut by selected polygons	is used to edit coverage area of DEM

Menu items	Function
Cut null edges	is used to remove edge areas of DEM, that include blank cells
Transpose	is used to transpose DEM from the left coordinate system to the right one and vice versa
Transform to another coordinate system	allows to change coordinate system of DEM
Shift	is used to parallel displacement of DEM by axes
Split DEM into sheets	allows to split DEM into sheet to save DEM in parts in separate files
Merge DEMs	allows to merge DEMs, that have overlap area
Accuracy control	is used to perform accuracy control DEM creation using various data
Convert to points...	allows to convert DEM cells to pickets with specified simplifying
Export...	contains menu items used to perform export of DEM to different formats
Import...	contains items used to import DEM from files with `grd`, `asc`, `tif`, `dem`, `mtw`, `dt1`, `dt2`, `img`, `pix`, `hgt` extensions
Batch import	is used to simultaneous import of multiple DEM
Load lidar data	is used to load and use laser scanning data (see detailed description in the 'LIDAR Data processing' User Manual)

7.2. General information

DEM (Digital Elevation Model) – is digital cartographic representation of the earth's surface in the form of a regular grid of height values.

A source data for DEM creation is the following data, taken individually or in combination:

- TIN (Triangulated Irregular Network);
- regular or irregular pickets (point objects);
- vector objects (see the *Vector objects creation* chapter in the 'Vectorization' User Manual).

DEM is displayed in 2D-window, and also could be opened to view in 3D-window or in 2D-window of stereopair in stereo mode.

DEM in stereo mode may be displayed inaccurately.

To view information about DEM select **Information** item in the *Manager*. The system displays information window which includes the following information about the layer:

- layer name;
- full path to layer file;
- presence of data and changes in the layer;
- possibility to change the layer;
- coordinates of DEM border nodes, including Z-coordinate;
- ground sample distance;
- DEM size (number of cells);
- layer coordinate system.

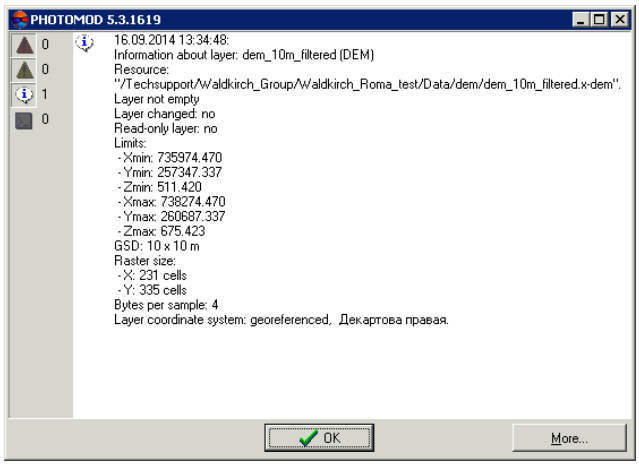


Fig. 70. Information about DEM layer

7.3. DEM creation

7.3.1. DEM creation by TIN

The system provides possibility to create DEM using irregular spatial triangles network (TIN). Perform the following actions to do this:

1. Create or load source TIN.
2. Choose **DEM › Build DEM › From TIN** (**Ctrl+N, D**). The **DEM by TIN creation parameters** window opens.

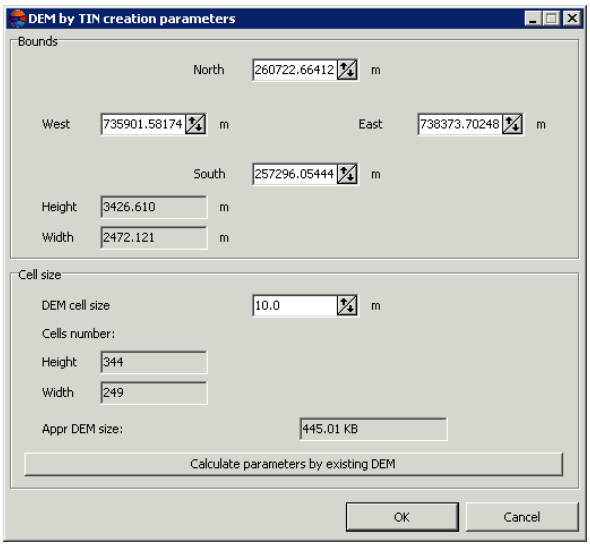


Fig. 71. DEM by TIN creation parameters

3. Specify DEM borders in the **North**, **West**, **East**, **South** fields. In the **Height** and **Width** fields the system displays calculated size of DEM border in meters.

   Default values are coordinates of rectangle corners, which generate TIN creation area.

4. In the **Cell size** specify the **DEM cell size** in meters to define cell size of output DEM. The **Cells number**, calculated considering specified cell size, displays in the **Height** and **Width** fields. Also the **Approximately DEM size** in megabytes displays in the appropriate field.

   DEM cell size should be comparable to average distance between points of base TIN layer. If less cell size is used, the time of DEM creation and size of output file increase, but accuracy is not.

5. [optional] It is possible to use parameters of existed DEM to calculate parameters of output DEM. To do that, click the **Calculate parameters by existing DEM** button and choose DEM file in active profile resources.

6. Click OK. Specify DEM file name, define a folder in active profile resources and click **Save** (see also Section 7.9 and DEM load). The system starts DEM creation operation. After that the system creates new layer in the *Manager* and shows a message about operation successful or unsuccessful completion.

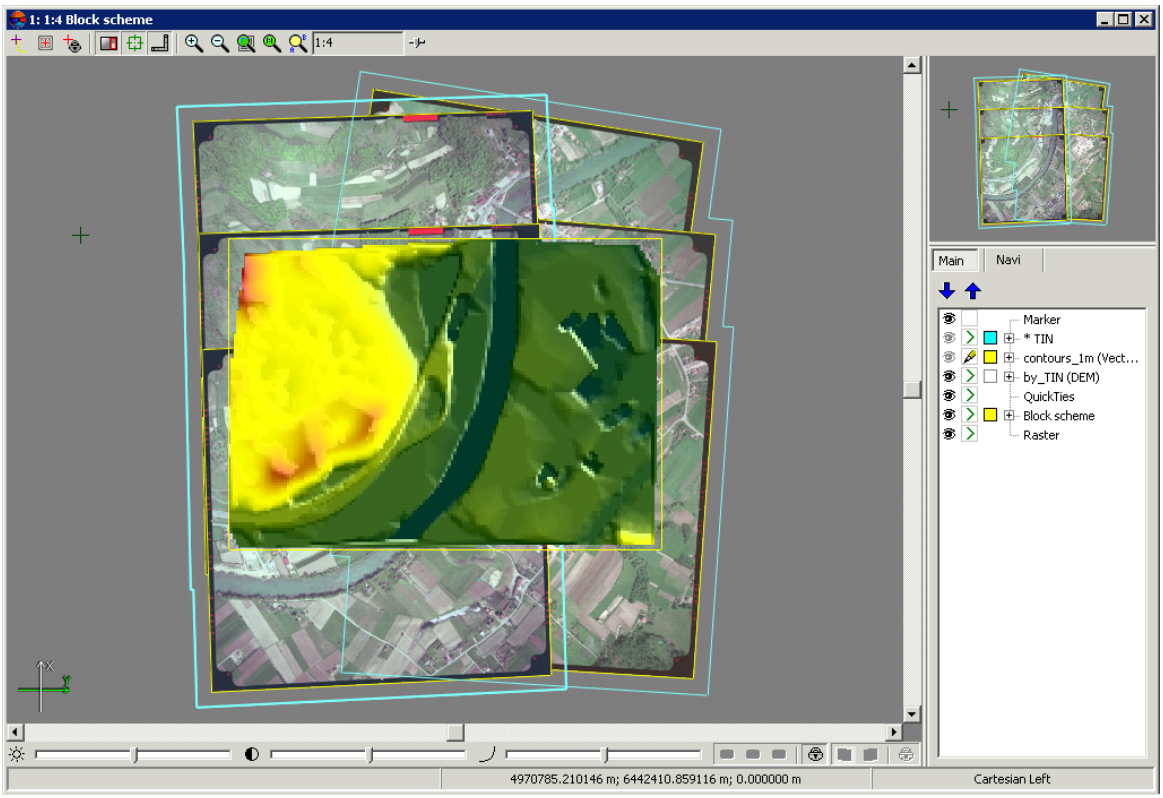


Fig. 72. DEM built from TIN

When marker is moving over DEM, the status panel in 2D-window displays XYZ-coordinates of DEM points.

The system allows to recreate DEM after changing of TIN base layer. To do this use the **DEM › Rebuild by TIN** menu item.

7.3.2. DEM creation using regular pickets

The system provides possibility to create DEM using a set of pickets created in regular grid nodes or imported from external files. Perform the following actions to do this:

1. Create or load a layer with pickets.

2. Select **DEM › Build DEM › From regular points...** The **Parameters of DEM building by regular pickets** parameters window opens.

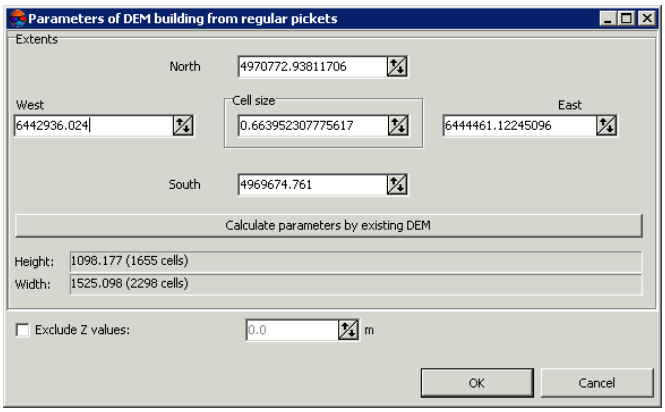


Fig. 73. DEM creation using regular pickets

3. Specify DEM borders in the **North**, **West**, **East**, **South** fields. In the **Height** and **Width** fields the system displays calculated size of DEM border in meters, as well as number of DEM nodes.

   Coordinates of border pickets of source layer are default values.

4. Specify **Cell size** in meters to change size of output DEM cell.

   DEM cell size should be commensurate with the average distance between pickets. If less cell size is used, the time of DEM creation and size of output file increase, but accuracy is not.

5. [optional] It is possible to use parameters of existed DEM to calculate parameters of output DEM. To do that, click the **Calculate parameters by existing DEM** button and choose DEM file in active profile resources.

6. [optional] In order to exclude pickets with specified elevation from creation operation, set the **Exclude Z values** checkbox on and input the parameters in meters.

7. Click OK. Specify DEM file name, define a folder in active profile resources and click **Save** (see also Section 7.9 and DEM load). The system starts DEM creation operation. After that the system creates new layer in the *Manager* and shows a message about operation successful or unsuccessful completion.

7.3.3. DEM creation using pickets

The system provides possibility to create DEM using pickets (point vector objects located on the terrain relief surface). Perform the following actions to do this:

1. Create or load a layer with pickets.

2. Select **DEM › Build DEM › From points...**. The **Parameters of DEM building from pickets** parameters window opens.

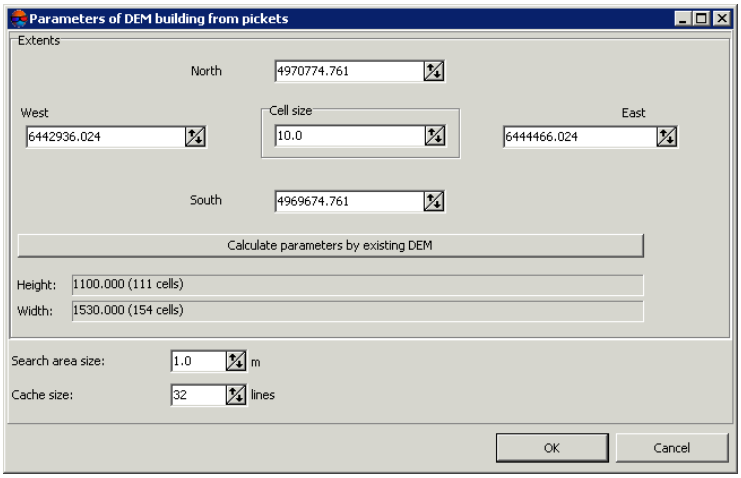


Fig. 74. Parameters of DEM creation from pickets

3. Specify DEM borders in the **North**, **West**, **East**, **South** fields. In the **Height** and **Width** fields the system displays calculated size of DEM border in meters, as well as number of DEM nodes.

Coordinates of border pickets of source layer are default values.

4. Specify **Cell size** in meters to change size of output DEM cell.

DEM cell size should be commensurate with the average distance between pickets. If less cell size is used, the time of DEM creation and size of output file increase, but accuracy is not.

5. [optional] It is possible to use parameters of existed DEM to calculate parameters of output DEM. To do that, click the **Calculate parameters by existing DEM** button and choose DEM file in active profile resources.

6. Specify **Search area size** in meters to change 'sliding window' size (in meters on terrain).

'Sliding window' method – means moving of some frame over set of pickets from node to node of DEM considering elevations and weight of pickets falling into the 'sliding' frame. A value of DEM cell (elevation), located in the middle of 'sliding window', is define as the sum of heights of the pickets in the 'sliding window' depending on their weight.

Cell size of output DEM should not exceed a size of the 'sliding window', specified in the **Search area size** field.

7. Specify **Cache size** parameter in lines of search for pickets in a specified search area by cell size.

   Cache size is used to define a number of lines of search area.

8. Click OK. Specify DEM file name, define a folder in active profile resources and click **Save** (see also Section 7.9 and DEM load). The system starts DEM creation operation. After that the system creates new layer in the *Manager* and shows a message about operation successful or unsuccessful completion.

7.3.4. DEM creation using CSV-file

The system provides possibility to create DEM using data obtained from CSV-file.

It is possible to use points coordinates from CSV-file only if points coordinates are recorded using the template `Name X Y Z`.

To create DEM by CSV-file perform the following actions:

1. Select **DEM › Build DEM › From CSV file data...** The **Parameters of DEM building by CSV file** parameters window opens.

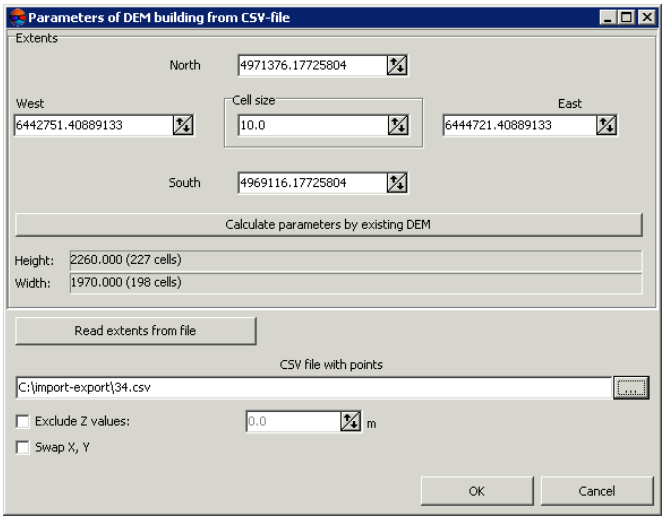


Fig. 75. DEM creation using CSV-file data

2. Specify DEM borders in the **North**, **West**, **East**, **South** fields. In the **Height** and **Width** fields the system displays calculated size of DEM border in meters, as well as number of DEM nodes.

   Coordinates of border pickets included to source CSV-file are default values.

3. [optional] To define output DEM border from CSV-file data click the **Read extents from file** button.

4. Specify **Cell size** in meters to change size of output DEM cell.

   DEM cell size should be commensurate with the average distance between pickets. If less cell size is used, the time of DEM creation and size of output file increase, but accuracy is not.

5. [optional] It is possible to use parameters of existed DEM to calculate parameters of output DEM. To do that, click the **Calculate parameters by existing DEM** button and choose DEM file in active profile resources.

6. Click the ... button and select CSV-file to be used for DEM creation.

7. [optional] In order to exclude pickets with specified elevation from creation operation, set the **Exclude Z values** checkbox on and specify the value in meters.

8. [optional] In order to swap coordinate system (left/right), set the **Swap X, Y** checkbox on. At that the coordinate system do not change.

   If the **Swap X Y** checkbox is cleared, the vector objects will be imported in the right coordinate system. Otherwise, in the left one.

9. Click OK. Specify DEM file name, define a folder in active profile resources and click **Save** (see also Section 7.9 and DEM load). The system starts DEM creation operation. After that the system creates new layer in the *Manager* and shows a message about operation successful or unsuccessful completion.

7.3.5. DEM creation using smooth model

The system provides possibility to build a DEM using smooth model created by objects of selected vector layers. When using smooth model the system creates smoothed DEM with specified accuracy.

To create DEM by smooth model perform the following actions:

1. Select **DEM › Build DEM › Smooth model...**. The **Parameters of DEM building by smooth model** window opens.

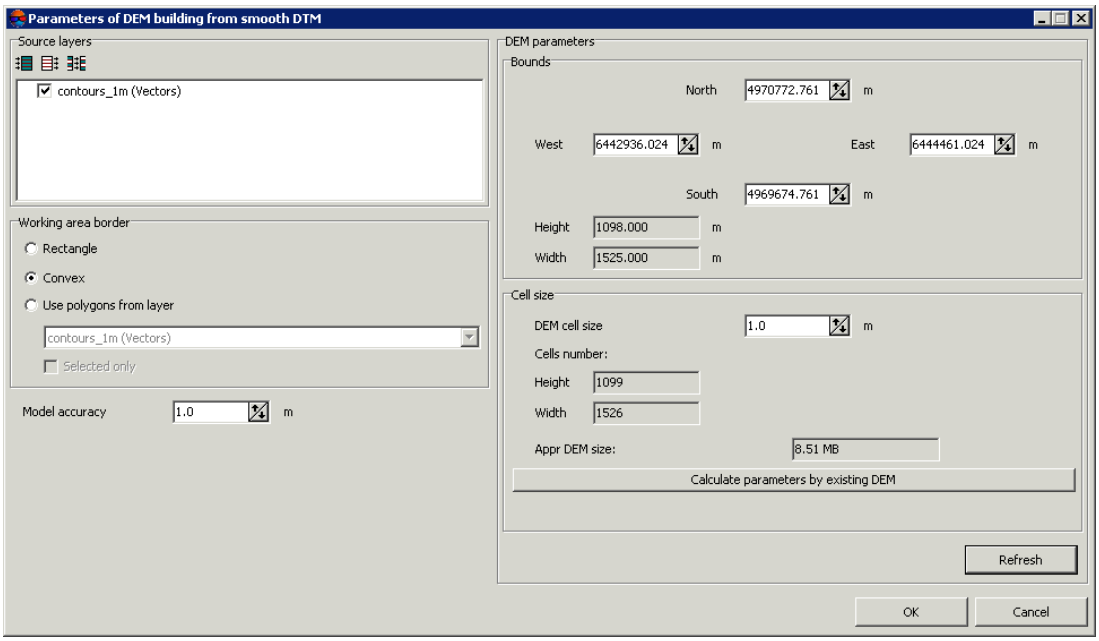


Fig. 76. Parameters of DEM creation using smooth model

2. In the **Source layers** section set the checkboxes on next to base point layers used for DEM creation.

   Click the button to select all layers, click the button to unselect all layers, or click the button, invert layers selection.

3. In the **Working area border** layer specify border type of working area:

   - **Rectangle** – used to create rectangular border;
   - **Convex** – used to create border using edge pickets of selected layer;
   - **Use polygons from layer** – used to create a border from the layer, which polygons are used as a border for DEM creation area. Set the **Selected only** checkbox on, to use just selected polygons as a border, otherwise, a DEM will be created considering all polygons of the layer.

4. In the **Model accuracy** field input acceptable deviation of DEM cells elevation values from source vector layer in meters.

   The **Model accuracy** parameter should be specified based on errors values obtained during creation of vector objects using correlator in stereomode.

5. Specify DEM borders in the **North**, **West**, **East**, **South** fields. In the **Height** and **Width** fields the system displays calculated size of DEM border in meters.

6. In the **Cell size** section specify **DEM cell size** in meters to change a size of output DEM cell. In the **Height** and **Width** fields the system displays nodes number calculated with specified DEM cell size. There is also displayed the **Appr DEM size** in megabytes.

   DEM cell size should be commensurate with the average distance between pickets. If less cell size is used, the time of DEM creation and size of output file increase, but accuracy is not.

7. [optional] It is possible to use parameters of existed DEM to calculate parameters of output DEM. To do that, click the **Calculate parameters by existing DEM** button and choose DEM file in active profile resources.

8. [optional] To reset to default parameters, click the **Refresh** button.

9. Click OK. Specify DEM file name, define a folder in active profile resources and click **Save** (see also Section 7.9 and DEM load). The system starts DEM creation operation. The new layer is created in the *layer manager*.

7.3.6. Creation of dense DTM

The system allows to create a dense DTM, which cell size coordinates to one pixel of image.

To perform dense DTM creation the system provides the following workflow:

1. DTM creation for specified area.

2. Main processing: apply filter of buildings and vegetation to created DTM and/or restore empty cells.

3. [optional] Additional processing – median and/or smoothing filtering.

   If intermediate data does not removed after the processing is complete, the system provides possibility to start separate steps (filtering and smoothing) without without re-calculation of the original DEM.

To create a dense DTM perform the following actions:

1. Create a grid to define DEM creation area.

   Since coordinates calculation is performed in each pixel of the selected area, the grid nodes are not considered. The grid is used only as a border of DTM creation area.

2. Select **DEM › Build DEM › Dense DTM...**. The **Parameters of create dense DTM** window opens.

If a grid not was not created in advance, the **Grid properties** window opens, that allows to create a grid for the entire images block.

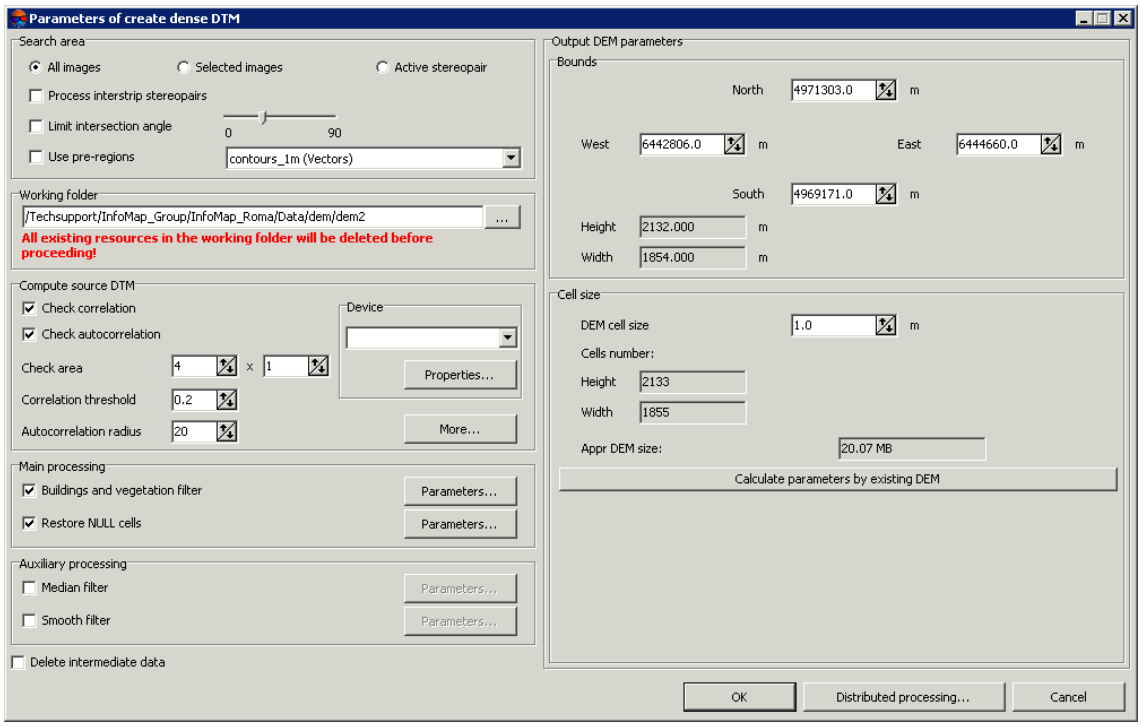


Fig. 77. Parameters of dense DTM creation

3. In the **Search area** section specify one of the areas of searching:

   - **All images** – allows to select all images of a block;
   - **Selected images** – allows to select block images highlighted in 2D-window;
   - **Active stereopair** – allows to select images of a stereopair opened in active 2D-window.

   All selected stereopairs should be included to adjusted part of the block (see the 'Block adjustment' User Manual). Otherwise, the system will fail to create DTM or will create it incorrectly. In the first case the system displays error message, in the second one – it is not possible to match calculated pickets to project coordinate system.

4. [optional] Specify additional parameters of search area:

   - set the **Process inter-strip stereopairs** checkbox on to calculate DTM on inter-strip stereopairs during UAS projects processing, and also for processing images, which do not require high creation accuracy;

   Pickets calculation is performed by default for strip stereopairs.

- set the **Limit intersection angle** checkbox on to filter blunders by Z on images with intersection angle of small or null value, and define a value of minimal angle using the slider;
- set the **Use pre-regions** checkbox on and select vector layer with bordering polygons.

5. In the **Working folder** section click the ... button and select **empty** folder in active profile resources to save output DEM.

Before operation is performed all data from selected folder is removed.

6. [optional] In the **Compute source DTM** section define parameters of correlator work:
   - the **Check correlation** and **Check autocorrelation** checkboxes are on in the system by default, that allows to check correlation and auto-correlation values whether they meet specified thresholds;

The filtering is performed using check of correlation threshold by auto-correlation radius.

It is not recommended to change pickets filtering parameters.

   - in the **Check area** fields specify a size of search matrix;
   - specify **Correlation threshold** – minimal accepted correlation coefficient value for matched pixels of images;
   - if there are multiple video adapters are installed on the workstation, select the CPU from the drop-down list or NVIDIA CUDA processor (GPU) for calculations. Click the **Properties** button to view video adapter's properties, if they are allowed by the device;

Using NVIDIA CUDA significantly reduces the processing time. See detailed information on the web-site http://www.nvidia.ru/object/cuda_gpus_ru.html.

   - click the **More** button to define additional parameters. The **Correlation parameters** window opens.

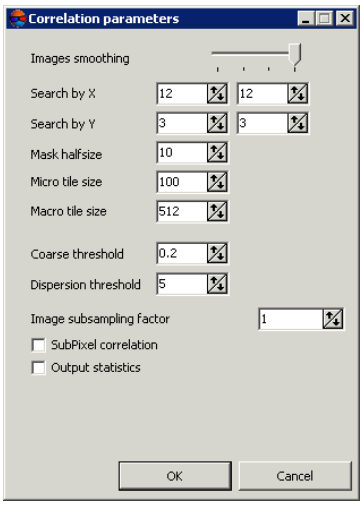


Fig. 78. Window of additional correlator parameters

Parameter	Description
Images smoothing	the slider used to define a level of images smoothing
Search by X	search area size by X with respect to the initial approximation calculated considering orientation data
Search by Y	search area size by Y with respect to the initial approximation calculated considering orientation data
Mask halfsize	a value of half of linear size of rectangular correlation mask in pixels by X and Y axes
Micro tile size	area size in microtile used to match pixels
Macro tile size	size of image area
Coarse threshold	a value of correlation coefficient during microtiles matching
Dispersion threshold	a value of contrast during matching of images pixels
Image subsampling factor	used level of image pyramid for points search
Subpixel correlation	a checkbox to consider subpixel correlation during search for corresponding points
Output statistics	a checkbox to display information about dense DEM creation in progress window

7. In the **Main processing** section the **Buildings and vegetation filter** checkbox is set on by default to exclude from processing those pickets, that were placed on buildings roofs or on crowns of trees. Set the checkbox off to not to use the filter, or click the **Parameters** button to configure filtering parameters, described in the Section 7.4.1.

8. [optional] In the **Main processing** section the **Restore null cells** is set on by default. Set the checkbox off to not to use the filter, or click the **Parameters** button to configure parameters. The **Parameters of filling nulls cells in DEM** window opens.

9. [optional] In the **Additional processing** section specify the following filtering parameters:

   - set the **Median filter** checkbox to use median filtering during DEM creation and specify **Parameters** of filter usage;

   - set the **Smooth filter** checkbox to use smooth filtering during DEM creation and specify **Parameters** of filter usage.

10. [optional] To remove processing data after the calculation is completed set the **Delete intermediate data** checkbox on.

    It is not recommended to remove intermediate data up to final creation of DEM. During removing of intermediate data it is impossible to start separate steps (filtering or smoothing) without DEM re-calculation.

11. In the **Output DEM parameters** section specify DEM boundaries in the **North**, **West**, **East**, **South** fields. In the **Height** and **Width** fields the system displays calculated size of DEM border in meters.

12. Specify **Cell size** in meters to change size of output DEM cell. In the **Height** and **Width** fields of the **Cell size** section the system displays nodes number calculated with specified DEM cell size. There is also displayed the **Appr DEM size** in megabytes.

    The value of the **DEM cell size** parameter should not be less than average pixel size (GSD). If less cell size is used, the time of DEM creation and size of output file increase, but accuracy is not.

13. [optional] The system provides possibility to calculate output DEM parameters using parameters of the existing DEM. To do this click the **Calculate parameters by existing DEM** button and select file with DEM to be used for calculation in active profile resources.

14. Click OK to start operation of dense DEM creation in usual mode.

    In order to perform DEM using distributed processing, do the following:

    During DEM distributed creation an area of output DEM is divided into rectangular sheets with a specified overlap, which are calculated in the individual tasks. When the last task is performed the system executes merging of all acquired sheets of the DEM.

1. Change settings and run the distributed processing server/client (see the '*Distributed processing*' chapter in the 'General information about system' User Manual).

2. Click the **Distributed processing** button. The **DEM distributed processing** window opens.

Prior to configure parameters of distributed processing it is necessary to specify parameters of using the buildings and vegetation filter. The settings window is opened automatically if the parameters are not specified.

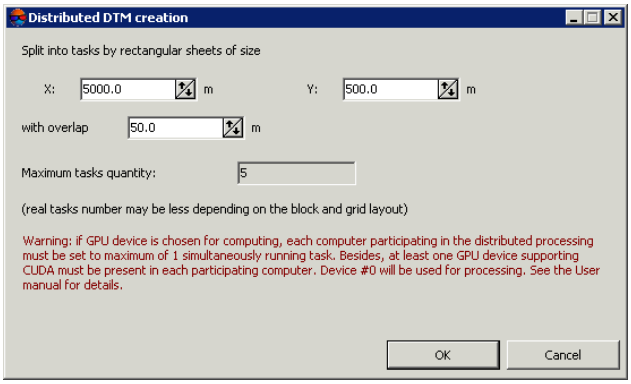

Distributed DTM creation
Split into tasks by rectangular sheets of size
X:
5000.0
m
Y:
500.0
m
with overlap
50.0
m
Maximum tasks quantity:
5
(real tasks number may be less depending on the block and grid layout)
Warning: if GPU device is chosen for computing, each computer participating in the distributed processing must be set to maximum of 1 simultaneously running task. Besides, at least one GPU device supporting CUDA must be present in each participating computer. Device #0 will be used for processing. See the User manual for details.
OK
Cancel

Fig. 79. Parameters of dense DTM creation in distributed processing mode

3. Set the parameters **Split into tasks by rectangular sheets of size by X** and **by Y** in meters, considering number of used computers.

It is recommended to set one task per one computer.

Maximal number of created tasks is calculated automatically considering specified sheet size.

If the GPU device is used to create DEM, it is necessary the specify **Maximum tasks quantity** not more than **one** for **each computer, involved in processing**.

It is necessary to have at least one graphics device available that supports CUDA, for **each** computer used for processing.

4. [optional] Specify the **with overlap** parameter value less than the default value to speed up the processing.

It is highly not recommended to decrease a value during processing of images acquired for terrain with big elevation differences .

A grid step should be much smaller than sheets overlap area, otherwise, there is the probability of getting area without DEM in the corners of sheets and after merging.

5. Click OK. Distributed processing tasks are created and the system shows a message about number of created tasks.

Depending on the size of creation area the operation can take a long time.

7.3.7. Creation of dense DSM

To create Digital Surface Model (DSM) with deformating method perform the following:

1. Setup and run the distributed processing server/client (see the '*Distributed processing*' chapter in the 'General information about system' User Manual).

2. Create a grid to define DEM creation area.

Since coordinates calculation is performed in each pixel of the selected area, the grid nodes are not considered. The grid is used only as a border of DTM creation area.

3. Select **DEM › Build DEM › Dense DSM... (deformating method)**. The **Create dense DSM** window opens.

If a grid not was not created in advance, the **Grid properties** window opens, that allows to create a grid for the entire images block.

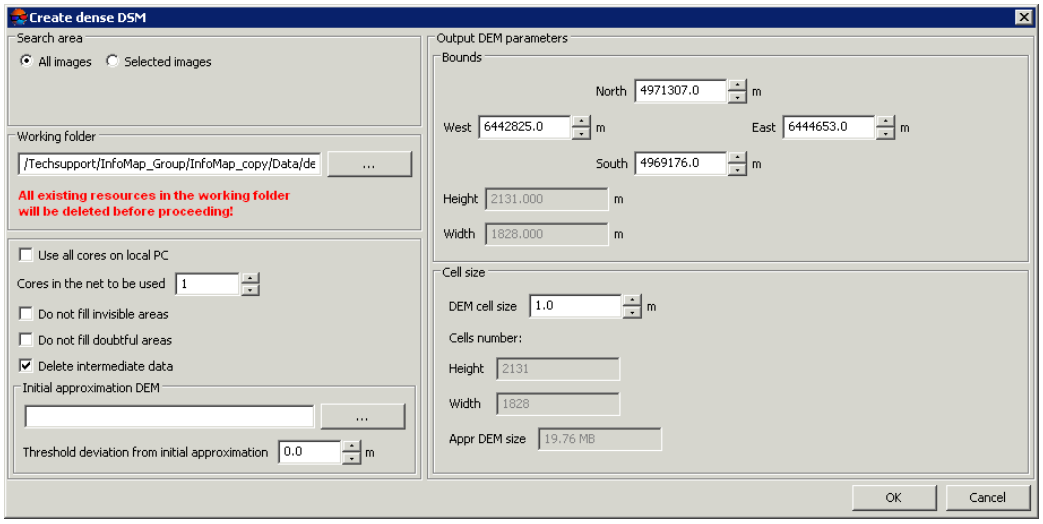


Fig. 80. Dense DSM creation parameters

4. In the **Search area** section specify one of the areas of searching:

   - **All images** – allows to select all images of a block;
   - **Selected images** – allows to select block images highlighted in 2D-window.

5. In the **Working folder** section click the ... button and select **empty** folder in active profile resources to save output DEM.

   Before operation is performed all data from selected folder is removed.

6. Choose the number of used cores:

   - set the **Use all cores on local PC** checkbox on to split process to tasks according to quantity of available cores (1 core – 1 task);

     Quantity of available cores is a quantity of cores on local PC.

   - set the **Cores in the net to be used** for DEM creation manually in the appropriate field.

     General quantity of tasks is calculated automatically.

7. Define the following parameters:

   - the **Do not fill invisible areas** checkbox allows to exclude areas, where ground is invisible (e.g., shadow from the house);
   - the **Do not fill doubtful areas** checkbox allows to exclude areas with low correlation coefficient, errors or homogeneous vast areas (dense forest) and so on.

8. [optional] To remove processing data after the calculation is completed set the **Delete intermediate data** checkbox on.

   It is not recommended to remove intermediate data up to final creation of DEM. During removing of intermediate data it is impossible to start separate steps (filtering or smoothing) without DEM re-calculation.

9. In the **Output DEM parameters** section specify DEM boundaries in the **North**, **West**, **East**, **South** fields. In the **Height** and **Width** fields the system displays calculated size of DEM border in meters.

10. Specify **Cell size** in meters to change size of output DEM cell. In the **Height** and **Width** fields of the **Cell size** section the system displays nodes number calculated with specified DEM cell size. There is also displayed the **Appr DEM size** in megabytes.

The value of the **DEM cell size** parameter should not be less than average pixel size (GSD). If less cell size is used, the time of DEM creation and size of output file increase, but accuracy is not.

11. Click OK. Operation of dense DEM creation starts.

7.3.8. DEM creation in batch mode

The system provides possibility to create DEM in batch mode. Batch mode allows to create multiple files with TIN and DEM simultaneously and to save them to specified folder.

For DEM batch creation the source vector resources are divided into sheets using specified parameters. For each vectors sheet the system creates TIN, used for DEMs creation, and then they are merged to generate a single output DEM.

Perform the following actions to execute batch processing:

1. Select **DEM › Build DEM › Batch DTM creation**. The **Create batch DTM – Step 1 of 2: Source data**.

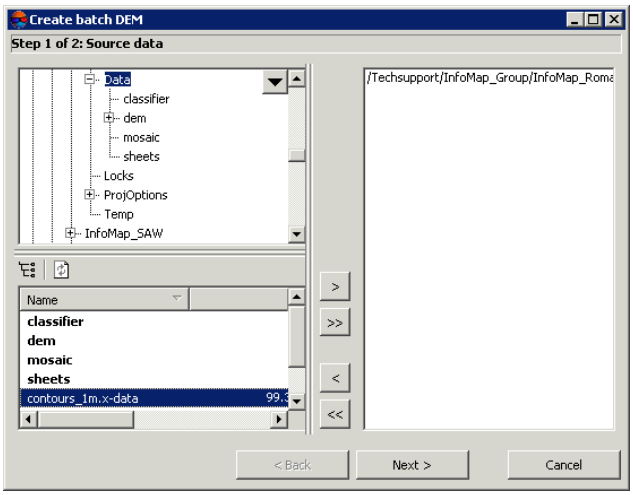


Fig. 81. Creating a list of the initial vector resources for batch DTM creation

2. Choose a folder containing vector objects in resources tree.

The button allows to display all available resources in nested files. The button allows to refresh part of the window with resources.

The button allows to show a list with 10 recently selected resources.

3. Select vector file in the list and click the > button to add a layer.

The >> and << buttons allow to add to the list or remove from it all added files with vector objects, the < button allows to remove selected file from the list

4. Repeat the 2 and 3 steps to add next files with vector objects.

5. Click the **Next** button. The **Create batch DTM – Step 2 of 2: Options**.

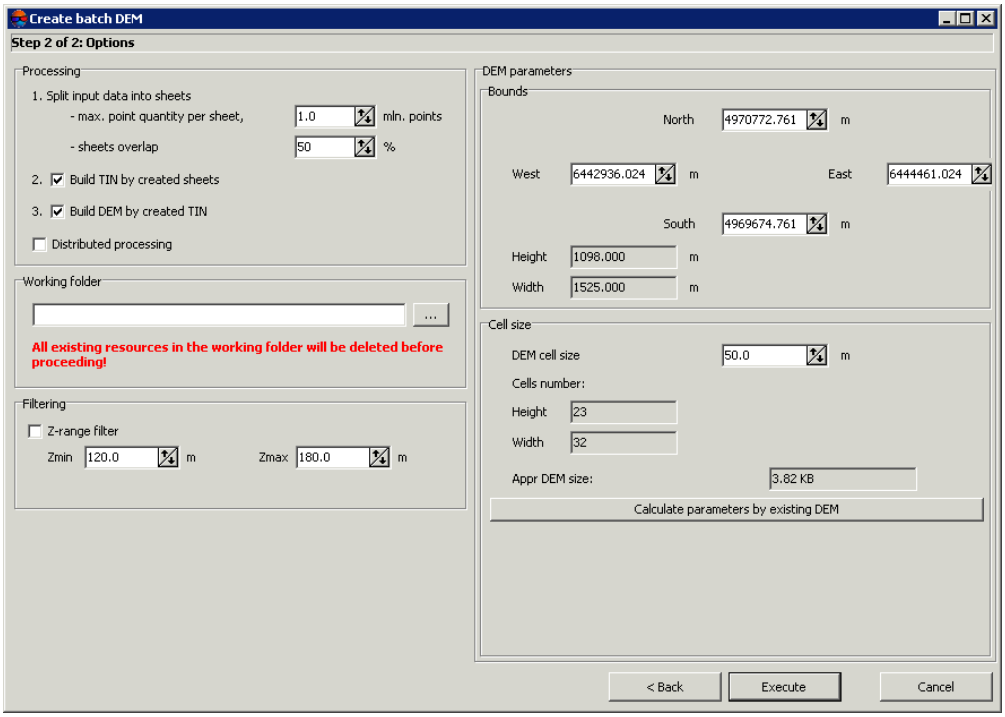


Fig. 82. Parameters of batch DTM creation

6. In the **Processing** section define tasks to be performed during DEM creation:

   1. **Split input data into sheets** – specify **max.point quantity per sheet** in millions of points, and specify a size of **sheets overlap** in percent.

      Specified parameters are considered by the system to define a number of sheets for splitting, so in specified target folder the system creates vector files with `x-data` extension and *sheet_sheet №* names.

   2. **Build TIN by created sheets** – (is set on by default) is used to create TIN by vector objects layers, divided into sheets. During execution of this task in the specified target folder the system creates files with `x-tin` extensions and *sheet_sheet №* names, that correspond to vector files.

   3. **Build DEM by created TIN** – (is set on by default) is used to create a single DEM by the whole TIN, generated using obtained sheets.

The task is performed in two steps: DEM is created for each sheet by TIN, after that all created DEMs are merged into single one. Thus in the output folder a file with `x-dem` extension is created.

7. [optional] Set the **Distributed processing** checkbox on to use distributed processing feature for batch DEM generation.

8. In the **Working folder** section define **empty** folder in active profile resources, where to place output data.

Before operation is performed all data from selected folder is removed.

9. [optional] In the **Filtering** section set the **Z-range filter** checkbox on to apply filtering to vectors and/or TIN. During filtering the system removes points and vertices of polylines/polygons (including TIN vertices), which Z-coordinate is out of range, specified in the **Zmin** and **Zmax** fields in meters.

10. In the **DEM parameters** section specify DEM boundaries in the **North**, **West**, **East**, **South** fields. In the **Height** and **Width** fields the system displays calculated size of DEM border in meters.

11. Specify **Cell size** in meters to change size of output DEM cell. In the **Height** and **Width** fields of the **Cell size** section the system displays nodes number calculated with specified DEM cell size. There is also displayed the **Appr DEM size** in megabytes.

DEM cell size should be commensurate with the average distance between pickets. If less cell size is used, the time of DEM creation and size of output file increase, but accuracy is not.

12. [optional] It is possible to use parameters of existed DEM to calculate parameters of output DEM. To do that, click the **Calculate parameters by existing DEM** button and choose DEM file in active profile resources.

13. Click the **Execute** button. The system displays a message about removing of all data in specified output folder. After that the system creates new DEM layer in the *Manager*.

    When the **Distributed processing** checkbox is set on, perform the following actions:

    1. Change settings and run the distributed processing server/client (see the '*Distributed processing*' chapter in the 'General information about system' User Manual).

    2. Click the **Execute** button. The **DEM distributed processing** window opens.

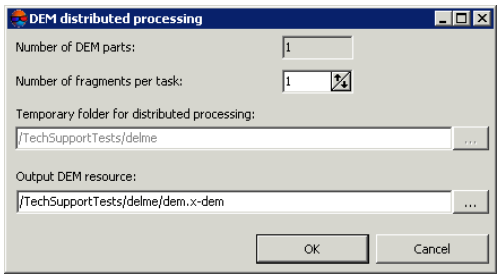


Fig. 83. Parameters of distributed batch DTM creation

3. The **number of DEM parts**, which the DEM is split on during distributed processing, is calculated automatically and depends on the size of the matrix. It is recommended to specify **Number of fragments per task** assuming one task per one computer.

4. Select the **Temporary folder for distributed processing** for temporary data storing.

5. Define the output file name and path.

6. Click OK. Distributed processing tasks are created and the system shows a message about number of created tasks.

7.4. DEM filtering

7.4.1. Buildings and vegetation filter

The system allows to use filtering of buildings and vegetation of DEM. Such filtering includes calculation of base pickets with specified coefficient of DEM subsampling. The operation includes deletion, correction or recognizing points which have fallen on buildings, trees, cars, in pits, etc. during automatic pickets calculation, as well as filtering of accident spikes. After the filtering output DTM contains just terrain relief model.

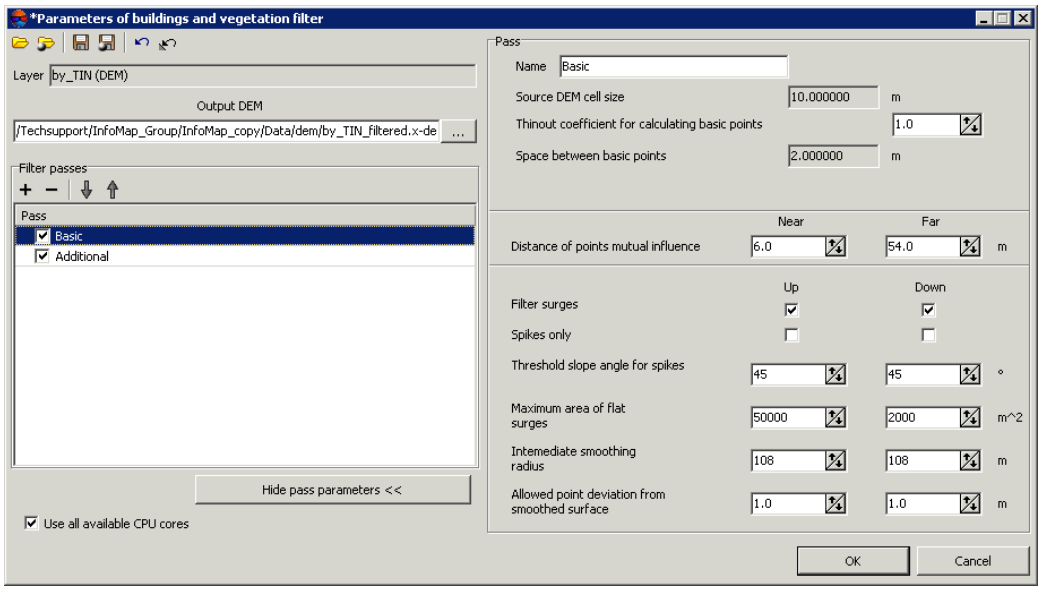


Fig. 84. Buildings and vegetation filtering from DEM

The **Buildings and vegetation filter** window allows to specify filtering parameters and setup a number of filter passes. The **Layer** field displays a name of active vector layer. The window contains a standard toolbar.

Button	Function
	allows to load a script of filter work from `*.x-filter` resource located out of active profile resources
	allows to load a script of filter work from `*.x-filter` resource located in active profile resources
	allows to save a current script of filter work to `*.x-filter` resource located out of active profile resources
	allows to save a current script of filter work to `*.x-filter` resource located in active profile resources
	allows to cancel all changes made to a script
	allows to go back to a standard script that includes two passes with default settings (regardless of which script has been loaded)

To filter buildings and vegetation on DEM perform the following actions:

1. Make active a DEM layer.

2. Select **DEM › Buildings and vegetation filter**. The **Buildings and vegetation filter** window opens.

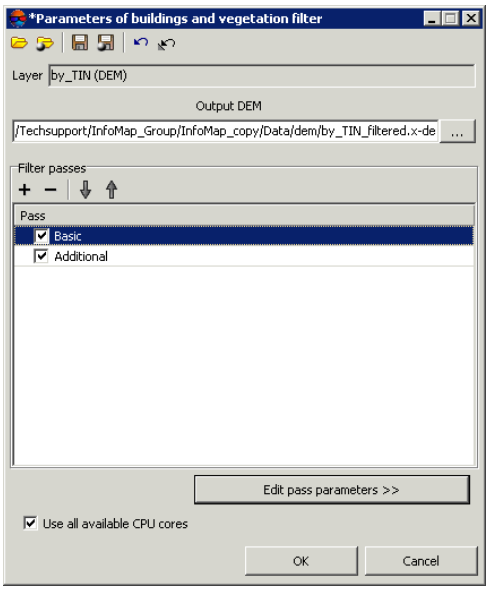


Fig. 85. Buildings and vegetation filtering from DEM

3. In the **Output DEM** section click the ... button and specify name and target folder of output DTM in active profile resources.

4. [optional] Specify a number of filter passes:

   - by default it is used two passes with standard parameters – **Basic** and **Additional**. Set off checkboxes to change number of passes;
   - the + button allows to add a new pass of filter;
   - the − button allows to remove selected pass of filter;
   - the ↓ and ↑ buttons allow to move selected pass up/down.

5. Select a name of pass and click the **Edit pass parameters** button to display and configure the following parameters of filter pass:

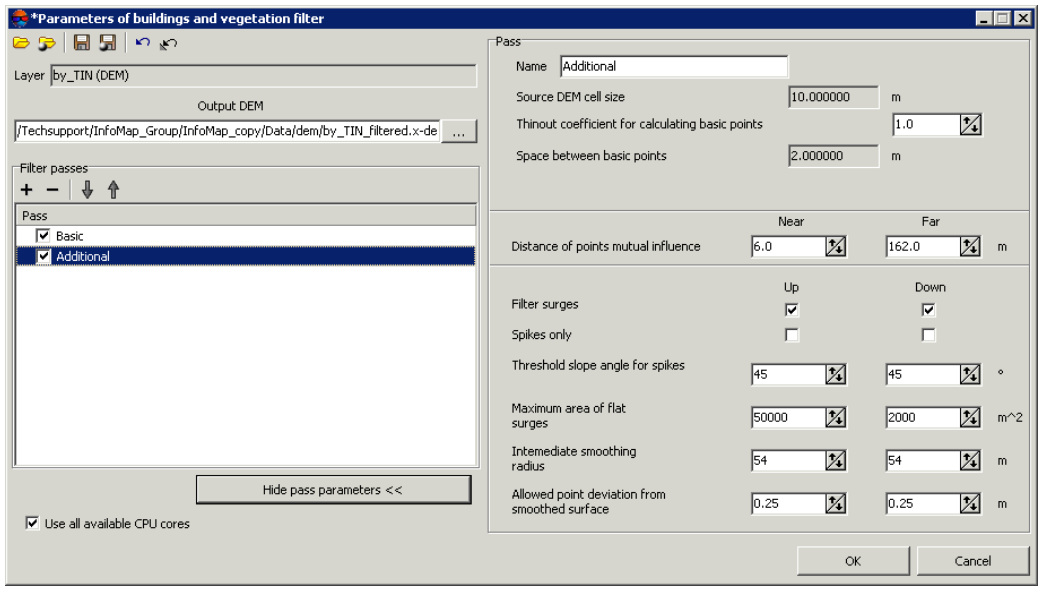


Fig. 86. Configuration of buildings and vegetation filter

- [optional] to change a filter pass name, input new name to the **Name** field;
- the **Source DEM cell size** field shows DEM cell size in meters before filter applying;
- specify **Thinout coefficient for calculating basic points** by DEM. From the specified value and the **Source DEM cell size** parameter depends the **Space between basic points** parameter in meters;
- specify **Near** and **Far distance of mutual points influence** in meters to define the radius of the circle, in which the values of pickets are analysed for errors. For near distance (minimal circle radius) it is recommended to set a value of 3-5 values of the average distances between pickets;
- specify the following filtering parameters **Up** (above surface) and/or **Down** (below surface):
  - **Filter surges** – allows to select pickets to be filtered: pickets on surface (high objects) and/or pickets below surface (pits);

    For filter operating it is necessary to set at least one checkbox on.

  - **Spikes only** – is used to reject just sharp spikes above/below surface, that are defined by values of the **Threshold slope angle for spikes** and **Distance of points** parameters. The rest of parameters are not considered;

In an angle between three points exceeds a value of the **Threshold slope angle for spikes** parameter, then the surge is called *spike*.

- **Threshold slope angle for spikes** – allows to specify angle of slope in relation to selected measuring plane (above and/or below surface) to define sharp spikes;
- **Maximum area of flat surges** – allows to define maximal area of plane surges (above/below surface) – points groups, that form smooth surfaces and are distant from some plane of neighbour points. Generally, the points are located on buildings roofs, and are lying on the same plane;

The filter is not applied to plane surface, which area exceeds the specified value.

- **Intermediate smoothing radius** – allows to specify a sphere radius (above/below surface), that defines a level of intermediate smoothing of surface;
- **Allowed point deviation from from smoothed surface** – allows to specify criterion, according which the filter is applied to all pickets with elevations that differ from smoothed surface more then on specified value.

The button allows to go back to a standard script that includes two passes with all default settings (regardless of which script has been loaded).

6. [optional] To use all cores of used computer, the **Use all available CPU cores** checkbox is set on in the system by default. It is necessary to set the checkbox off to use only one core.
7. Click OK. The system starts DEM filtering operation. When the filtering operation is completed the system displays information message about number of used basic points and pickets being filtered.

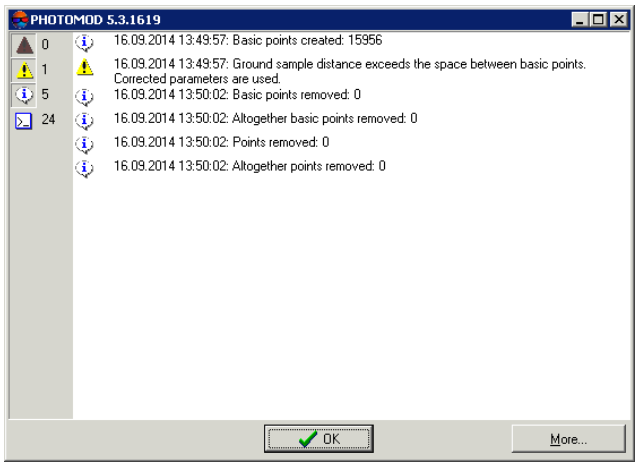

PHOTOMOD 5.3.1619
0
1
5
24
16.09.2014 13:49:57: Basic points created: 15956
16.09.2014 13:49:57: Ground sample distance exceeds the space between basic points. Corrected parameters are used.
16.09.2014 13:50:02: Basic points removed: 0
16.09.2014 13:50:02: Altogether basic points removed: 0
16.09.2014 13:50:02: Points removed: 0
16.09.2014 13:50:02: Altogether points removed: 0
OK
More...

Fig. 87. Filtering results

7.4.2. Median filter

The system allows to use median filtering of DEM cells.

The *median filtering* contains the following sequence of actions:

1. Sequential scanning of DEM by mask with specified size. Size of scanning mask is defined by the **Aperture** parameter (in DEM cells).

2. The system replaces current cell elevation values by filtered values, if a difference between them is not more than specified **Threshold** in meters.

To apply median filtering perform the following actions:

1. Make active a DEM layer.

2. Select **DEM › Median filter**. The **Median filter parameters** window opens.

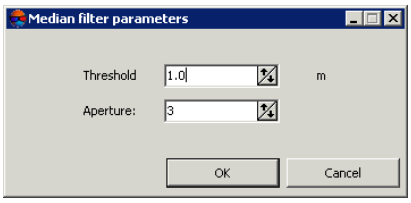


Fig. 88. Median filter parameters

3. In the **Threshold** filed specify a value of threshold in meters, which is used to compare difference value between current and filtered elevation.

4. In the **Aperture** field specify a size of scanning mask.

5. Click OK. After that current values of DEM cells are replaced by filtered values, if difference value between current and filtered elevation does not exceed specified **Threshold**.

7.4.3. Smooth filter

The system allows to use smooth filtering of DEM cells values.

To execute smooth filtering perform the following actions:

1. Make active a DEM layer.
2. Select **DEM › Smooth filter**. The **Smooth filter parameters** window opens.

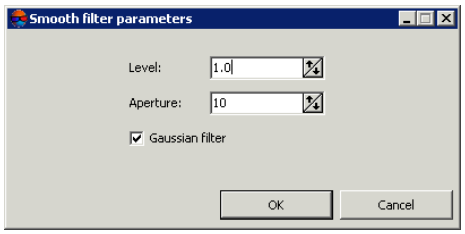


Fig. 89. Smooth filter parameters

3. In the **Level** field specify filtering coefficient value from 0 to 1.
4. In the **Aperture** field specify a size of scanning mask.
5. [optional] To apply Gauss smoothing algorithm the **Gaussian filter** checkbox is set on by default.
6. Click OK. After that current values of DEM cells elevation are replaced by filtered values.

7.5. Transformation of DEM coordinates

7.5.1. DEM transpose

The system provides opportunity to transpose DEM from the left coordinate system to the right one and vice versa. To do this use the **DEM › Transpose** menu item. Such a transformation results in swap of DEM rows and columns.

This feature is used, for example, if vector objects coordinates are incorrectly specified (see the chapter '*Transformation of objects coordinates*' in the 'Vectorization' User Manual).

7.5.2. Transformation of DEM coordinate system

The system provides opportunity to transform DEM from one coordinate system to another.

In order to transform DEM to another coordinate system perform the following actions:

1. Select **DEM › Transform to another coordinate system**. The **Conversion DEM to another coordinate system** window opens.

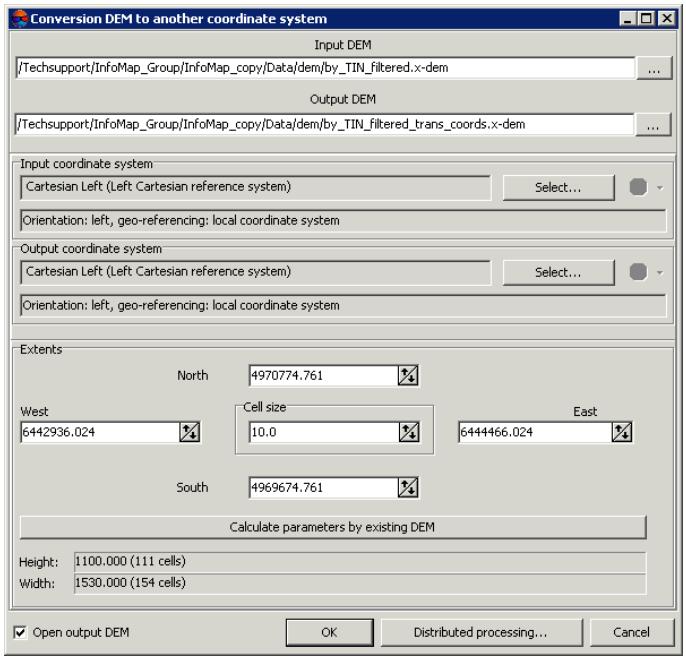


Fig. 90. Transform DEM to another coordinate system

2. Click the ... button in the **Input DEM** section and specify a source DEM file in active profile resources.

3. Click the ... button in the **Output DEM** section and specify DEM name and target folder in active profile resources.

4. In the **Initial coordinates system** section select actual Coordinate system of DEM (if it is different from project coordinate system). Perform the following actions to do this:

    1. Click the **Select...** button to specify initial coordinate system.

        Coordinate system is specified using one of the following ways:

        - **From DB** – from international or Russian coordinate system database (see*'Coordinate systems databases'* in the Project creation' User Manual);

- **From file** – allows to select coordinate system from files with `x-ref-system`, extension located out of active profile resources;
- **From resource** – from files with `x-ref-system` extension located in active profile resources, for example, to select coordinate system from another active profile project.

The system also allows to select coordinate system from a list of recently used coordinate systems.

2. [optional] When choosing coordinate system from database the **Coordinate system database** opens, which contains the list of coordinate systems.

To perform fast search for coordinate system, input the whole coordinate system name or its part to the **Find** input filed.

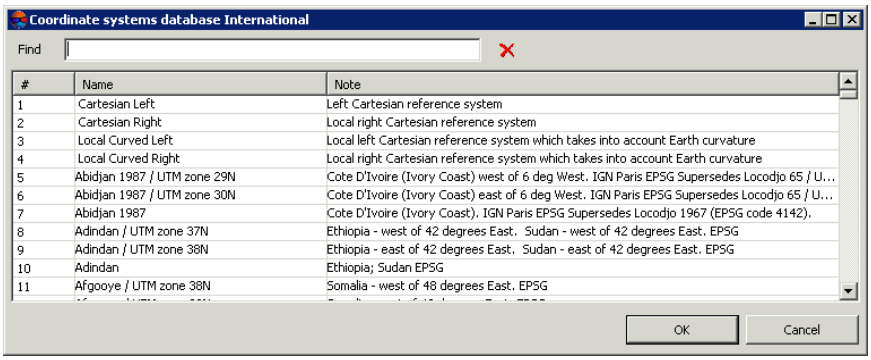

Coordinate systems database International
Find
Name Note
1 Cartesian Left Left Cartesian reference system
2 Cartesian Right Local right Cartesian reference system
3 Local Curved Left Local left Cartesian reference system which takes into account Earth curvature
4 Local Curved Right Local right Cartesian reference system which takes into account Earth curvature
5 Abidjan 1987 / UTM zone 29N Cote D'Ivoire (Ivory Coast) west of 6 deg West. IGN Paris EPSG Supersedes Locodjo 65 / U...
6 Abidjan 1987 / UTM zone 30N Cote D'Ivoire (Ivory Coast) east of 6 deg West. IGN Paris EPSG Supersedes Locodjo 65 / U...
7 Abidjan 1987 Cote D'Ivoire (Ivory Coast). IGN Paris EPSG Supersedes Locodjo 1967 (EPSG code 4142).
8 Adindan / UTM zone 37N Ethiopia - west of 42 degrees East. Sudan - west of 42 degrees East. EPSG
9 Adindan / UTM zone 38N Ethiopia - east of 42 degrees East. Sudan - east of 42 degrees East. EPSG
10 Adindan Ethiopia; Sudan EPSG
11 Afgooye / UTM zone 38N Somalia - west of 48 degrees East. EPSG
OK
Cancel

Fig. 91. Window used to select coordinate system from coordinate system database

3. [optional] To choose geoid to be used, click the button. Select proper type of geoid usage:
   - **No geoid**;
   - **EGM 96**.

The system allows to use the EGM2008 geoid. See more details in the Installation EGM2008 Geoid User Manual. After installation the geoid is displayed in the list.

5. In the **Output coordinate system** define target DEM coordinate system in which it is necessary to convert the DEM. To do this perform actions from step 4.
6. Specify DEM borders in the **North**, **West**, **East**, **South** fields. In the **Height** and **Width** fields the system displays calculated size of DEM border in meters, as well as number of DEM nodes.
7. Specify **Cell size** in meters to change size of output DEM cell.

8. [optional] It is possible to use parameters of existed DEM to calculate parameters of output DEM. To do that, click the **Calculate parameters by existing DEM** button and choose DEM file in active profile resources.

9. [optional] To open DEM after recalculation of coordinate system the **Open output DEM** checkbox is set on by default. Set the checkbox off to not load the created file in the project.

10. Click OK to convert DEM coordinate system.

    To use distributed computing for changing of DEM coordinate system, do the following:

    1. Change settings and run the distributed processing server/client (see the '*Distributed processing*' chapter in the 'General information about system' User Manual).

    2. Click the **Distributed processing** button. The **DEM distributed processing** window opens.

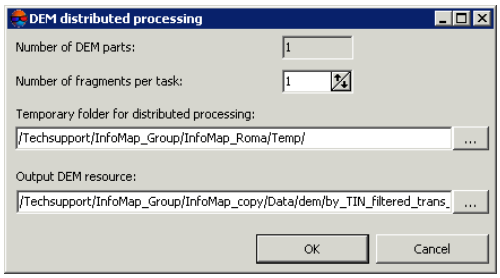


Fig. 92. DEM distributed processing

    3. The **number of DEM parts**, which the DEM is split on during distributed processing, is calculated automatically and depends on the size of the matrix. It is recommended to specify **Number of fragments per task** assuming one task per one computer.

    4. Select the **Temporary folder for distributed processing** for temporary data storing.

    5. Define the output file name and path.

    6. Click OK. Distributed processing tasks are created and the system shows a message about number of created tasks.

7.6. Null cells in DEM

7.6.1. General information

During automatic creation of regular relief model (DEM) some part of the cells can not be defined. Such DEM cells (’gaps’), which elevation is not known, are called *null cells*.

The system provides possibility to restore null cells of DEM using the following methods of null cells filling:

- linear interpolation method (see Section 7.6.2);
- smooth interpolation method (see Section 7.6.3);
- filling with constant value (see Section 7.6.4).

The system also allows to transform DEM cells with specified elevation to null cells.

Filling DEM null cells is also used to change input DEM cell size.

7.6.2. Filling null cells using linear interpolation

The system provides possibility to restore null cells of DEM using the linear interpolation method. Null cells filling using this method is performed by creating TIN from the DEM and calculation of null cells elevation using method of linear interpolation by created TIN. When using this method it is possible to change the DEM borders and cell size, as well as to remove ’sharp’ peaks of TIN.

To fill DEM null cells by linear interpolation method perform the following methods:

1. Select **DEM › Fill null cells › Linear interpolation...**. The **Parameters of filling null cells in DEM by linear interpolation** window opens.

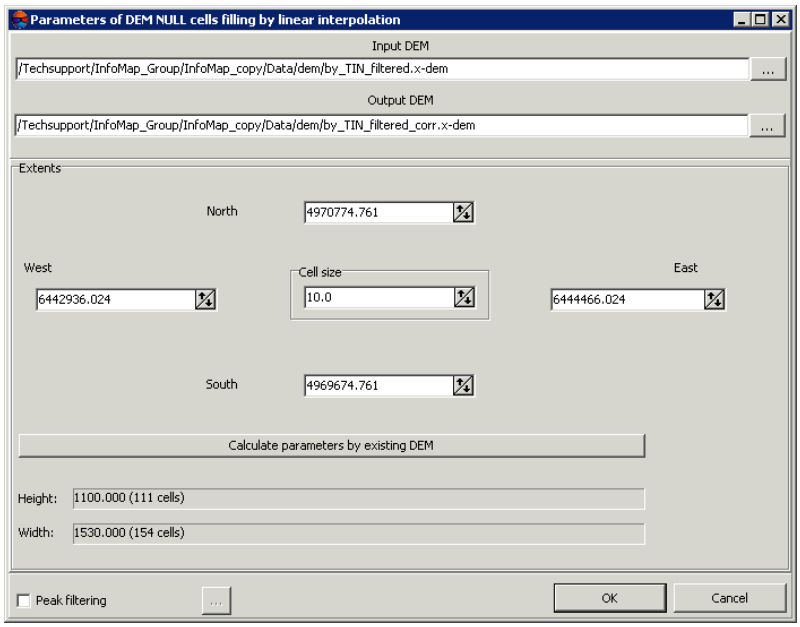


Fig. 93. Parameters of filling null cells using linear interpolation

2. In the **Input DEM** section click the ... button to select input DEM in active profile resources.

   The DEM loaded to the project is selected by default.

3. [optional] To define name and target folder of output DEM in active profile resources, click the ... button.

   By default the system suggests the *<input dem name>_corr.x-dem* name for output DEM and to place it to the folder, containing input DEM file.

4. In the **Extents** section specify DEM boundaries in the **North**, **West**, **East**, **South** fields. In the **Height** and **Width** fields the system displays calculated size of DEM border in meters, as well as number of DEM nodes.

5. Specify **Cell size** in meters to change size of output DEM cell.

6. [optional] It is possible to use parameters of existed DEM to calculate parameters of output DEM. To do that, click the **Calculate parameters by existing DEM** button and choose DEM file in active profile resources.

7. [optional] To remove sharp peaks in DEM cells with filled elevation values set the **Peak filtering** checkbox on and click the ... button. The **Parameters** window opens.

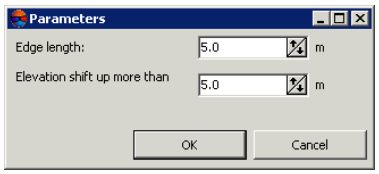


Fig. 94. Parameters of peak filtering

Input the following peak filtering parameters:

- **Edge length** – allows to specify maximal edge length in DEM cell (in meters);
- **Elevation shift up more than** – allows to specify maximal deviation of cell by elevation above adjacent cells (in meters).

8. Click OK. The system starts calculation of null cells elevation using linear interpolation method. After that the system creates new DEM layer in the *Manager* and shows a message about operation successful or unsuccessful completion.

7.6.3. Filling null cells using smooth interpolation

The system provides possibility to restore null cells of DEM using the smooth interpolation method. When applying the method of smooth interpolation the system evaluates the factor of input DEM thinning to obtain auxiliary DEM. An auxiliary DEM is used to calculate elevation values of null cells, and DEM cells with filled elevations are not re-calculated. Borders and cell size of output DEM are fully correspond to the boundary and the cell size of the input DEM.

This method is recommended for a sufficiently dense DEM.

To fill DEM null cells by smooth interpolation method perform the following methods:

1. Select **DEM › Fill null cells › Smooth interpolation...**. The **Parameters of filling null cells in DEM** window opens.

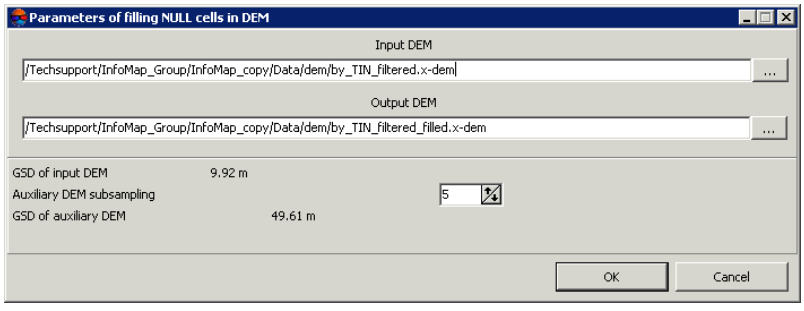


Fig. 95. Parameters of filling null cells using smooth interpolation

The window displays the **ground sample distance (GSD)** of input DEM in meters.

2. In the **Input DEM** section click the ... button to select input DEM in active profile resources.

   The DEM loaded to the project is selected by default.

3. [optional] To define name and target folder of output DEM in active profile resources, click the ... button.

   By default the system suggests the *<input dem name>_filled.x-dem* name for output DEM and to place it to the folder, containing input DEM file.

4. Input the **Auxiliary DEM subsampling**. After that the **GSD of auxiliary DEM** is calculated automatically.

5. Click OK. The system starts calculation of null cells elevation using smooth interpolation method. After that the system creates new DEM layer in the *Manager*.

7.6.4. Filling null cells by constant value

The system provides possibility to fill null cells of DEM by constant value.

To fill DEM null cells with constant value perform the following methods:

1. Select **DEM › Fill null cells › By constant value...**. The **Fill DEM null cells by constant value parameters** window opens.

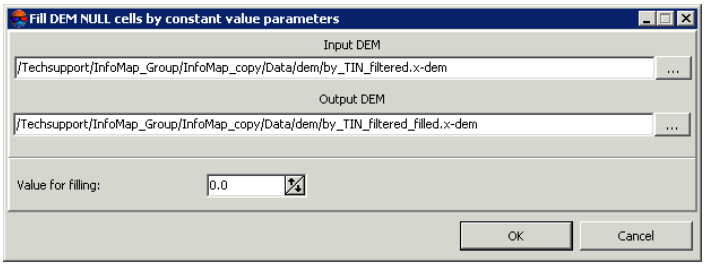


Fig. 96. Parameters of filling null cells of DEM by constant value

2. In the **Input DEM** section click the ... button to select input DEM in active profile resources.

   The DEM loaded to the project is selected by default.

3. [optional] To define name and target folder of output DEM in active profile resources, click the ... button.

By default the system suggests the *<input dem name>_filled.x-dem* name for output DEM and to place it to the folder, containing input DEM file.

4. Input the **Value for filling** of null cells.

5. Click OK. The system starts filling of null cells operation. After that the system displays a message about number of changed cells and creates new DEM layer in the *Manager*.

7.6.5. Converting cells to null

The system allows to convert DEM cells with specified elevation to null cells.

It is recommended to apply this method if there are errors after importing DEM.

In order to convert cells to null ones perform the following actions:

1. Select **DEM › Convert cells to null...**. The **Parameters of converting DEM cells to null values** window opens.

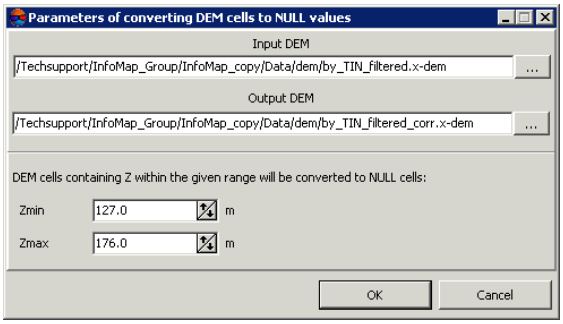


Fig. 97. Parameters of converting DEM cells to null values

2. In the **Input DEM** section click the ... button to select input DEM in active profile resources.

The DEM loaded to the project is selected by default.

3. [optional] To define name and target folder of output DEM in active profile resources, click the ... button.

By default the system suggests the *<input dem name>_corr.x-dem* name for output DEM and to place it to the folder, containing input DEM file.

4. Specify a range of DEM cells to be converted to null in the **Zmin** and **Zmax** fields in meters.

5. Click OK. After that all output DEM cells, which elevation falls within specified Z-range are converted into null cells.

7.7. DEM editing

7.7.1. DEM merging

The system provides possibility of merging (joining) of adjacent DEMs that have overlap.

To merge adjacent DEMs perform the following actions:

1. Select **DEM › Merge DEMs...**. The **DEM merging – Step 1 from 2:** window opens **Source data**.

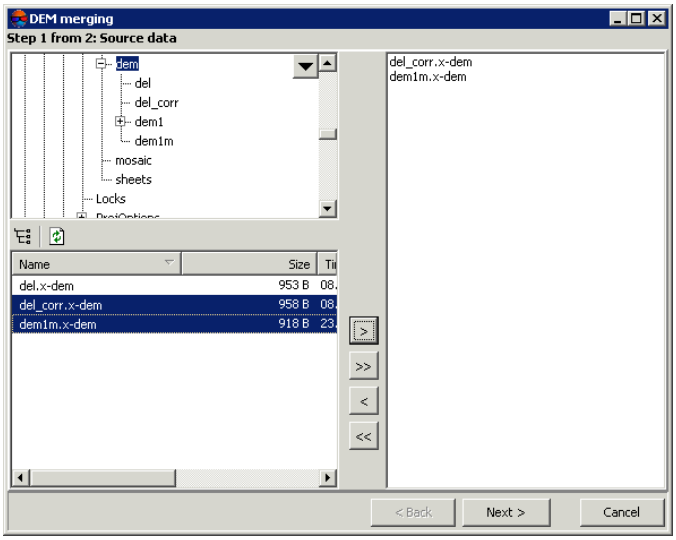


Fig. 98. Selection DEMs for merging

2. Choose a folder containing DEM in resources tree.

The button allows to display all available resources in nested files. The button allows to refresh part of the window with resources.

The button allows to show a list with 10 recently selected resources.

3. Select the DEM that has overlap with the adjacent one in the list, and click the > button to add the DEM to be merged.

The >> and << buttons allow to add to the list or remove from it all added DEMs, the > button allows to remove selected DEM from the list.

4. Repeat the 2 and 3 actions to add next DEMs.

It is necessary to choose for merging at least two files containing DEMs with overlap area.

5. Click the **Next** button. The **DEM merging – Step 2 from 2: Parameters** window opens.

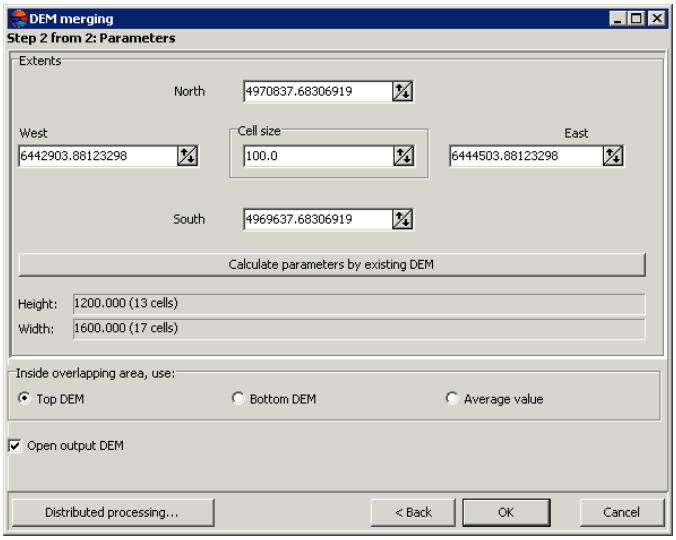


Fig. 99. Parameters of DEMs for merging

6. Specify DEM borders in the **North**, **West**, **East**, **South** fields. In the **Height** and **Width** fields the system displays calculated size of DEM border in meters, as well as number of DEM nodes.

7. Specify **Cell size** in meters to change size of output DEM cell.

8. [optional] It is possible to use parameters of existed DEM to calculate parameters of output DEM. To do that, click the **Calculate parameters by existing DEM** button and choose DEM file in active profile resources.

9. In the **Inside overlapping area, use** section select the DEM, which values should be used in the overlap area:

   - use data from top DEM (the first or the top DEM in the list, added for merging, in the **DEM merging – Step 1 from 2: Source data** window);

- use data from bottom DEM (the second or the bottom DEM in the list, added for merging, in the **DEM merging – Step 1 from 2: Source data** window);
- calculate and use average cells values of adjacent DEMs in overlap area.

10. [optional] By default the **Open output DEM** checkbox is set on that allows to open the out put DEM after the operation is completed. Set the checkbox off to not load the resulting DEM in the project.

11. Click OK to start DEM merging process.

    To use distributed computing for DEMs merging do the following:

    1. Change settings and run the distributed processing server/client (see the '*Distributed processing*' chapter in the 'General information about system' User Manual).

    2. Click the **Distributed processing** button. The **DEM distributed processing** window opens.

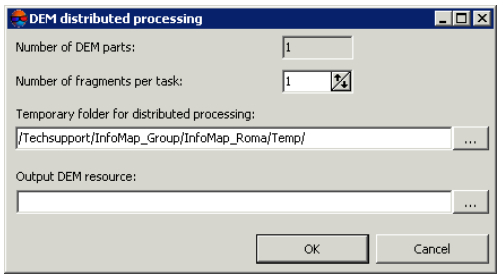


Fig. 100. DEM distributed processing

    3. The **number of DEM parts**, which the DEM is split on during distributed processing, is calculated automatically and depends on the size of DEMs. It is recommended to specify **Number of fragments per task** assuming one task per one computer.

    4. Select the **Temporary folder for distributed processing** for temporary data storing.

    5. Define the output file name and path.

    6. Click OK. Distributed processing tasks are created and the system shows a message about number of created tasks.

7.7.2. Change DEM parts elevation

The system allows to align selected part of DEM by elevation. In this case, an average elevation value of the DEM part is assigned to all cells of the DEM part outlined by selected polygon.

To align DEM part by elevation perform the following actions:

1. Create or load a vector layer (see the 'Vectorization' User Manual).
2. Create single or multiple polygons, that outline selected parts of the DEM, or load polygons layer to use them as boundaries.
3. Select at least one polygon that will be used as a boundary.
4. Choose **DEM › Set elevation in selected polygons**. The **Parameters of DEM elevation assigning in polygons** window opens.

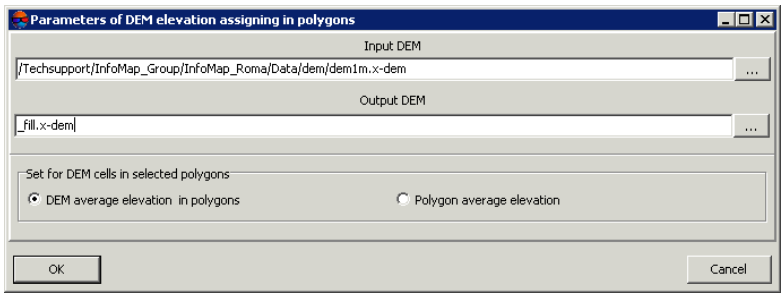


Fig. 101. Parameters of DEM elevation assigning in polygons

5. In the **Input DEM** section click the ... button to select input DEM in active profile resources.

   The DEM loaded to the project is selected by default.

6. [optional] To define name and target folder of output DEM in active profile resources, click the ... button.

   By default the system suggests the *<input dem name>_fill.x-dem* name for output DEM and to place it to the folder, containing input DEM file.

7. Specify value of DEM cells in each selected polygon:
   - **DEM average elevation in polygons**;
   - **polygon average elevation**.

8. Click OK. After that a specified average elevation value is assigned to all DEM cells inside selected polygons.

7.7.3. DEM shift

The system provides possibility to shift the whole DEM by one or more axes using parallel displacement.

In order to shift a DEM perform the following actions:

1. Load or create a DEM.

2. Select **DEM › Shift**. The **DEM shift** window opens.

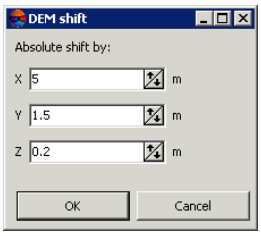


Fig. 102. Parameters of DEM shift

3. Specify shift value by axes in the **X**, **Y**, **Z** fields in meters.

It is possible to shift a DEM by one or all axes at once.

4. Click OK. After that all points of active layer DEM are shifted on specified value by selected axes with help of parallel displacement.

7.7.4. Split DEM into sheets

Splitting of DEM into sheet is available in the system. It is used to save DEM in parts in separate files.

To split DEM into sheets perform the following:

1. Load or create DEM.

2. Choose **Vectors › Open** to load existed sheets or create polygons to use as sheets.

3. Make *active* the DEM layer.

4. Choose **DEM › Split DEM by sheets**. The **Split DEM by sheets parameters** window opens.

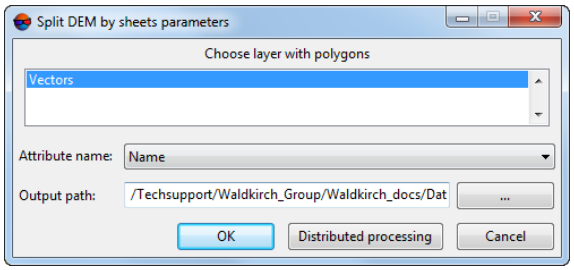


Fig. 103. Split DEM by sheets parameters

5. Click on the layer with polygons in the list.

6. Choose the **Attribute name** to use its values as filenames.

   If there is no chosen attribute for some sheets, these sheets can be passed or continuous numbering can be used as file names.

7. Choose the **Output path** for files.

8. Click OK to start the process.

   Perform the following to use distributed processing:

   1. Change settings and run the distributed processing server/client (see the '*Distributed processing*' chapter in the 'General information about system' User Manual).

   2. Click the **Distributed processing** button. The **Split DEM to sheets: distributed processing** window opens.

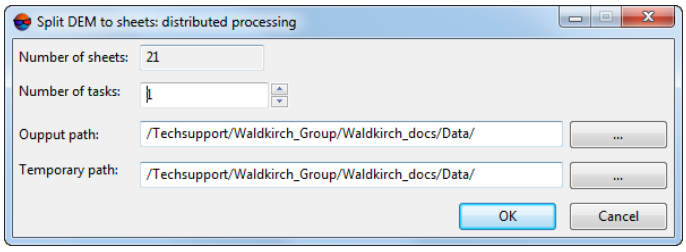


Fig. 104. Distributed processing parameters

   3. **Number of sheets** is calculated automatically and depends on number of polygons.

      It is trecommended to define **Number of tasks** based on "one task – one computer".

4. Choose the **Output path** for files.

5. Choose the **Temporary path** for temporary files.

6. Click OK. Distributed processing tasks are created and the system shows a message about number of created tasks.

As a result DEM splits into parts by polygons of chosen layer. Values of selected attribute are used as filenames.

7.7.5. DEM interpolating

The system supports local alignment of elevation values of DEM part, limited by a polygon on a vector layer.

In order to align a part of DEM perform the following actions:

1. Create a vector layer (see the ’Vectorization’ User Manual).

2. Create single or multiple polygons, that outline selected parts of the DEM, or load polygons layer to use them as boundaries.

3. Select at least one boundary polygon to be used for interpolation of cell elevation values inside it.

4. Choose **DEM › Interpolate elevation in selected polygons**. The **Parameters of DEM interpolation assigning in polygons** window opens.

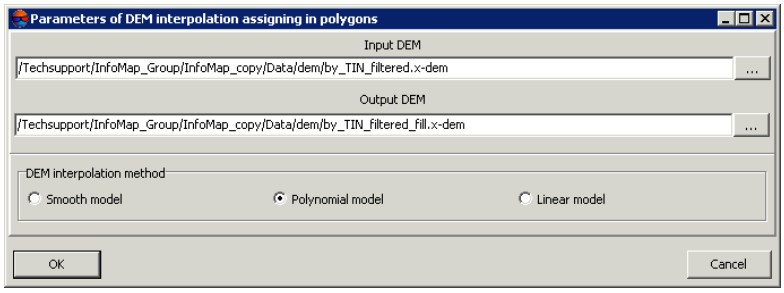


Fig. 105. Parameters of DEM interpolation assigning in polygons

5. In the **Input DEM** section click the ... button to select input DEM in active profile resources.

The DEM loaded to the project is selected by default.

6. [optional] To define name and target folder of output DEM in active profile resources, click the ... button.

By default the system suggests the *<input dem name>_fill.x-dem* name for output DEM and to place it to the folder, containing input DEM file.

7. Specify the **DEM interpolation method**:
   - **Smooth model**;
   - **Polynomial model**;
   - **Linear model**.
8. Click OK. After that cell elevation values inside selected polygons (one or several) is interpolated using specified method.

7.7.6. DEM cutting by polygons

The system allows to edit the area of DEM coverage.

In order to correct DEM area, perform the following actions:

1. Create vector layer and polygons in it, or open a layer with bordering polygons (see the 'Vectorization' User Manual).
2. Select one or more polygons, that will be used to cut the DEM.
3. Select **DEM › Cut by selected polygons**. The **Polygon cutting parameters** window opens.

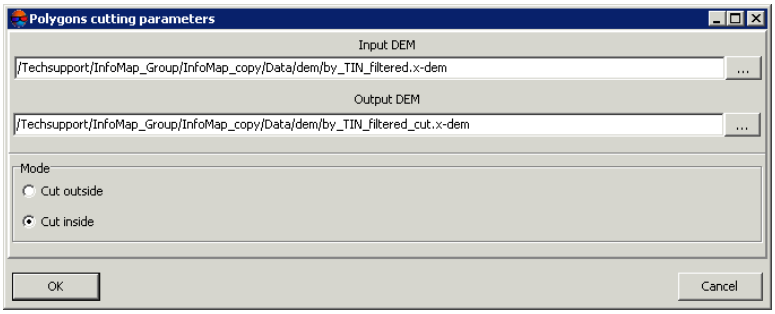


Fig. 106. Parameters of cutting by polygons

4. In the **Input DEM** section click the ... button to select input DEM in active profile resources.

The DEM loaded to the project is selected by default.

5. [optional] To define name and target folder of output DEM in active profile resources, click the ... button.

   By default the system suggests the *<input dem name>_cut.x-dem* name for output DEM and to place it to the folder, containing input DEM file.

6. Specify the **Mode** of DEM cutting:

   - **Cut outside** – allows to crop DEM outside of selected polygons area;
   - **Cut inside** – allows to cut DEM area located inside of selected polygons.

7. Click OK. After that resulting DEM obtained after editing is opened.

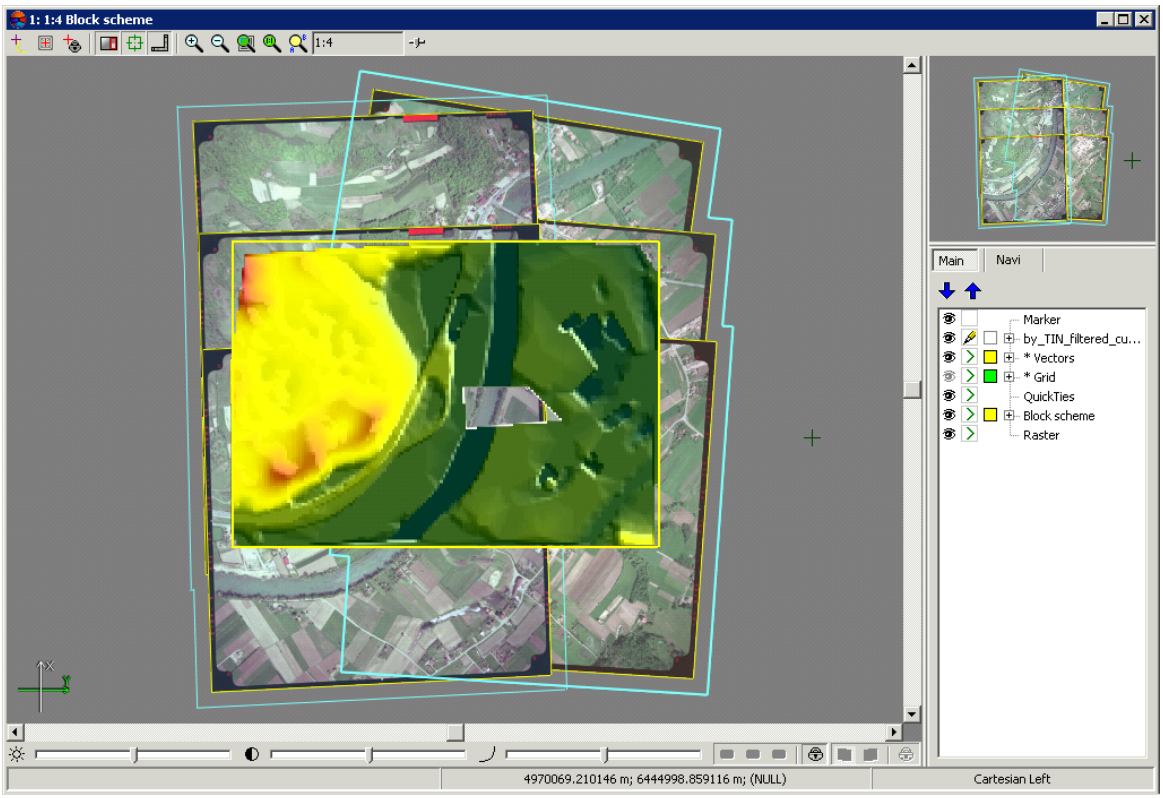


Fig. 107. DEM with areas cut inside polygons

Besides, the system allows to remove edge areas of DEM, that include null cells Perform the following actions to do this:

1. Select **DEM › Cut null edges**. The **Cut null edges** window opens.

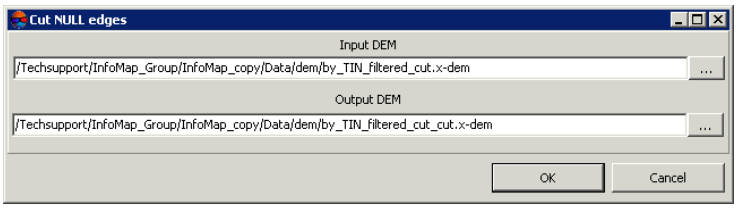


Fig. 108. DEM edges cutting

2. In the **Input DEM** section click the ... button to select input DEM in active profile resources.

   The DEM loaded to the project is selected by default.

3. [optional] To define name and target folder of output DEM in active profile resources, click the ... button.

   By default the system suggests the <*input dem name*>*_cut.x-dem* name for output DEM and to place it to the folder, containing input DEM file.

4. Click OK. After that the system opens a DEM without removed edges areas.

7.7.7. The 'DEM editing' toolbar

The system provides possibility of fast editing of DEM areas using selecting tools (see details in the *'Objects selection'* chapter of the 'Vectorization' User Manual). For this there is an additional toolbar **DEM editing** with buttons that allow to perform the following operations:

- – allows to cut DEM area located inside of selected polygon;
- – allows to set elevation of DEM cells inside of selected area;
- – allows to interpolate selected DEM area using smooth model;
- – allows to interpolate selected DEM area using polynomial model;
- – allows to interpolate selected DEM area using linear model.

To quickly edit a DEM area perform the following actions:

1. Load a DEM or make active a layer with DEM.
2. Click the button of the **DEM editing** additional toolbar, to define operation to be performed with DEM cells.

3. [optional] To specify elevation of selected DEM cells, set the elevation value in the filed to the right from the button.

4. Press and hold the **Shift** key and select area to be edited by mouse. The editing is performed automatically using parameters specified by default.

7.8. Accuracy control of DEM creation

7.8.1. Creation of difference DEM

The system provides possibility to create difference DEM using subtraction of one DEM from the other.

To create difference DEM perform the following actions:

1. Load at least two DEMs to be used for subtraction.

2. Select **DEM › Check accuracy › Build difference DEM**. The **DEMs comparing parameters** window opens.

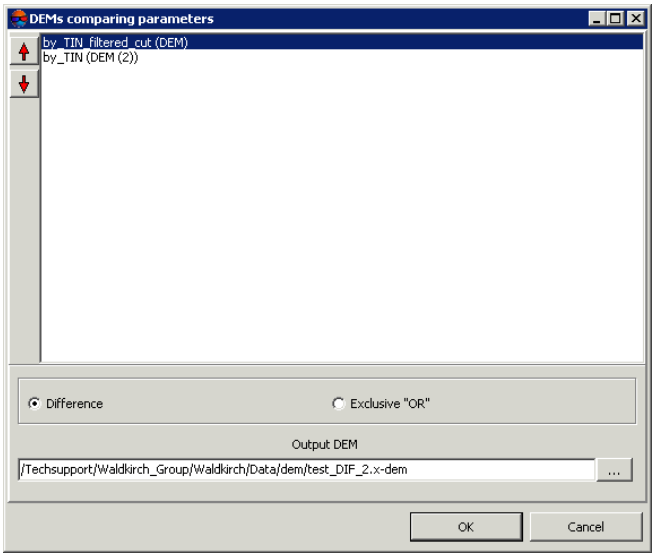


Fig. 109. Parameters of DEMs comparing

The list displays all loaded DEMs.

3. Select in the list a layer with DEM, from which the system will subtract. Use the and buttons to move it to the top of the list.

4. Select a way of resulting DEM creation:

   - **Difference** – if there are null values in the top DEM, the output DEM will contain null values in the same cells;

- **Exclusive 'OR'** – if there are null values in the top DEM, in the output DEM these null cells will be filled by values of the bottom DEM.

The final sign (positive or negative) of cells elevation in output DEM depends on sequence of subtracting of one DEM from the other. The sign of the cells elevation in null area is selected according to which of the DEMs from which is subtracted.

5. Select two DEMs in the list. To do this press and hold the **Ctrl** key and click the DEM list by mouse.

The DEM from which the system performs the subtraction should be higher in the list, than the subtrahend DEM.

6. In the **Output DEM** section click the ... button and specify name and path of output DTM in active profile resources.

7. Click OK. After that the system shows a message about operation successful or unsuccessful completion. In case of success the system creates new layer in the *Manager*.

For automatic positioning marker on active DEM surface the **Marker Z coordinate follows DEM in active layer** check box is on by default in the general system's parameters.

7.8.2. DEMs comparing

The system provides possibility of comparing of two DEMs for the same territory to analyse their discrepancies by Z axis.

To compare DEMs perform the following actions:

1. Load DEMs to be compared.

2. Select **DEM › Check accuracy › Compare DEMs**. The **DEMs compare parameters** window opens.

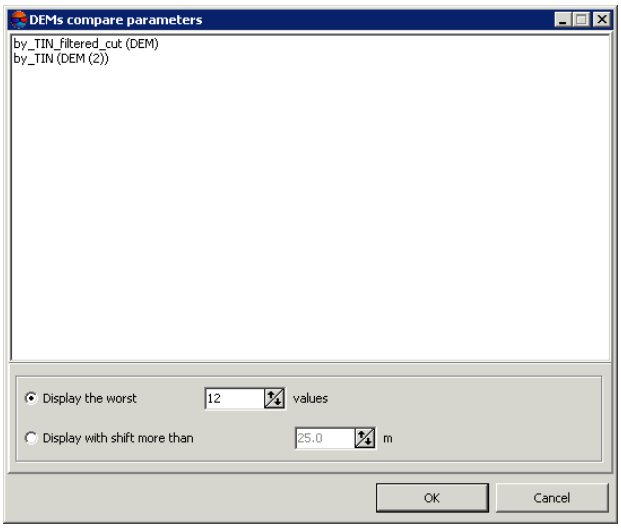


Fig. 110. Parameters of DEMs comparing

The list displays all loaded DEMs.

3. Set the following parameters:

   - **Display the worst..values** – allows to show a list of specified number of points with maximal error;

   - **Display with shift more than..m** – allows to show a list of points with discrepancy greater than specified.

4. Select two DEMs in the list. To do this press and hold the **Ctrl** key and click the DEM list by mouse.

5. Click OK. The system starts operation of selected DEMs comparing. When the operation is completed the **Compare DEMs** window opens.

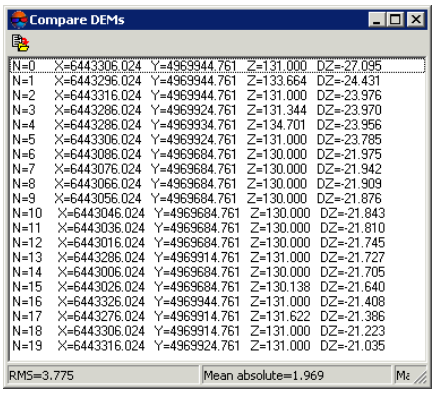


Fig. 111. Parameters of DEMs comparing

The window displays a list of points coordinates, where:

- `N` – is picket serial number;
- `X, Y, Z` – coordinates of erroneous picket;
- `DZ` – discrepancy by Z.

The status line of the window displays root mean square error (`RMS`) of discrepancies, `Mean absolute`, maximal discrepancy by Z found (`Max`), as well as `Average` discrepancy among points found.

The button allows to save the table to file with `csv` extension.

Double click the point record line by mouse to select the point in the DEM active layer.

7.8.3. Search for null cells

To perform analysis of DEM creation quality the system provides possibility of null cells search in created DEM.

It is not recommended to search for null cells when processing big data amounts.

To search for null cells in DEM perform the following actions:

1. Load or create a DEM.

   If multiple DEMs are loaded into project, the search is performed in the *active* DEM layer.

2. Select **DEM › Check accuracy › Search for null cells**. The **Null cells searching parameters** window opens.

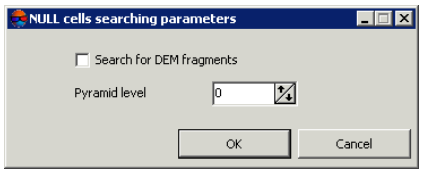


Fig. 112. Parameters of search for null cells

3. [optional] In DEM includes fragment that may contain null cells, set the **Search for DEM fragments** checkbox on.
4. Input the image **Pyramid level** for null cells search.

   Zero pyramid level corresponds to a source image.

5. Click OK. The system starts search for null cells operation. When the search is completed the system opens the **Null cells** window or displays a message about absence of null cells.

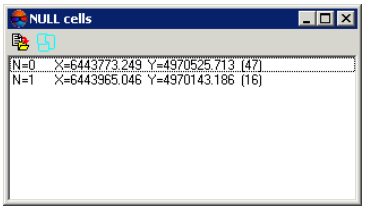


Fig. 113. Null cells list

The window displays a list of areas with null cells, where:

- `N` – is the area serial number;
- `X,Y` – coordinates of centers of null cells areas;
- `(31)` – number of null cells in area.

The button allows to save the list to file with `csv` extension.

To create a vector layer that contain points on borders of null cells area, click the button.

To edit DEM with null cells the system provides the following features:

- various methods of null cells filling (see Section 7.6);
- alignment of selected DEM fragment by elevation (see Section 7.6);
- vector objects editing and DEM rebuilding, when base vector layers are used for DEM creation.

7.8.4. DEM control by TIN

The system provides possibility to control created DEM by TIN. The control is performed by calculation of source TIN nodes discrepancies from created DEM.

To control created DEM perform the following actions:

1. Select **DEM › Check accuracy › Check against TIN**. The **Parameters of check DEM using TIN** window opens.

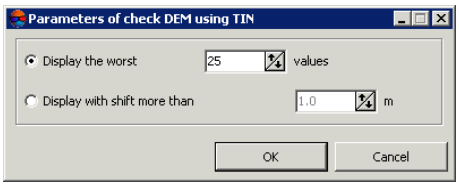


Fig. 114. Parameters of DEM accuracy control by TIN

2. Set the following parameters:

   - **Display the worst..values** – allows to show a list of specified number of TIN nodes with maximal error;
   - **Display with shift more than..m** – allows to show a list of points with discrepancy from DEM by Z greater than specified.

3. Click OK. The system starts operation of selected DEMs comparing. When the operation is completed the system opens the **Check DEM using TIN** window.

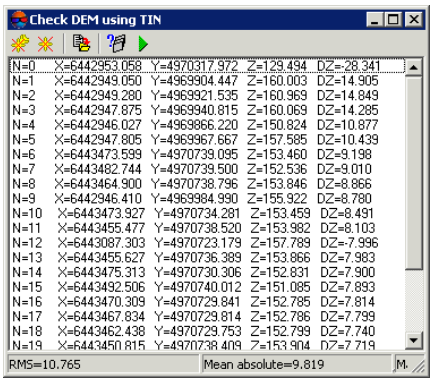


Fig. 115. Results of DEM control by TIN

The window displays a list of points coordinates, where:

- `N` – is serial number;
- `X, Y, Z` – coordinates of erroneous picket;
- `DZ` – discrepancy by Z between DEM and TIN.

The status line of the window displays root mean square error (`RMS`) of discrepancies, `Mean absolute`, maximal discrepancy by Z found (`Max`) on points found.

The button allows to save the list to file with `csv` extension.

Double click the point record line by mouse to select the point in the DEM active layer.

A list of erroneous nodes includes those located on all stereopairs of current project.

4. [optional] To change the list display parameters, click the button. In order to display changes in the list click the button.

5. To edit TIN using the list the system provides the following features:

   - to remove selected TIN node, click the button or press the **Delete** key;
   - to remove all nodes, where the error is more or equal to the error selected in the list, click the button.

6. Click the button, to re-build the DEM considering all changes.

7.8.5. DEM control by vector objects

The system provides possibility to control DEM creation quality by comparing DEM vertices coordinates with coordinates of vertices of vector objects layer. This vector objects layer should not be used as a base layer during this DEM or TIN creation.

DEM control by vector objects is needed to estimate accuracy of imported DEM or, vice versa, of vector objects by 'master' DEM.

During comparing vector objects with DEM the system compares Z-coordinates of pickets or vector objects vertices with elevation in corresponding points of the DEM.

To control DEM creation against vector objects perform the following actions:

1. Select **DEM › Check accuracy › Check against vector objects**. The **Parameters of check DEM by vectors** window opens.

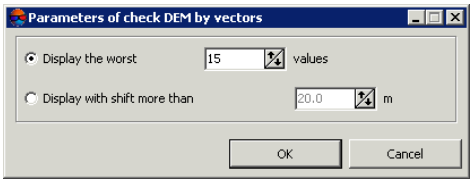


Fig. 116. Parameters of accuracy control by vectors

2. Set the following parameters:

   - **Display the worst..values** – allows to show a list of specified number of pickets with maximal error;
   - **Display with shift more than..m** – allows to show a list of points with discrepancy from DEM by Z greater than specified.

3. Click OK. The system starts operation of selected DEMs comparing. When the operation is completed the system opens the **Check DEM using vector objects** window.

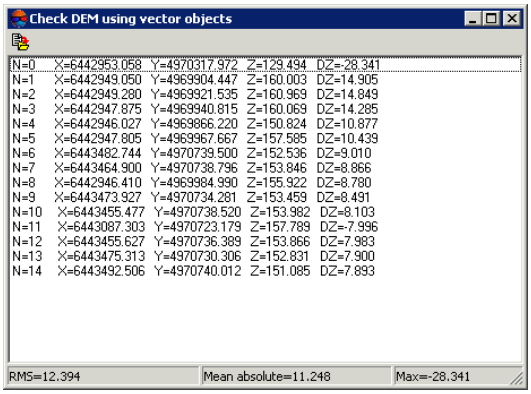


Fig. 117. Results of DEM control by vector objects

The window displays a list of points coordinates, where:

- `N` – is serial number;
- `X, Y, Z` – coordinates of erroneous picket;
- `DZ` – discrepancy by Z between DEM and vector objects vertices.

The status line of the window displays root mean square error (`RMS`) of discrepancies, `Mean absolute`, maximal discrepancy by Z found (`Max`) on points found.

The button allows to save the list to file with `csv` extension.

Double click the point record line by mouse to select the point in the DEM active layer.

A list of erroneous nodes includes those located on all stereopairs of current project.

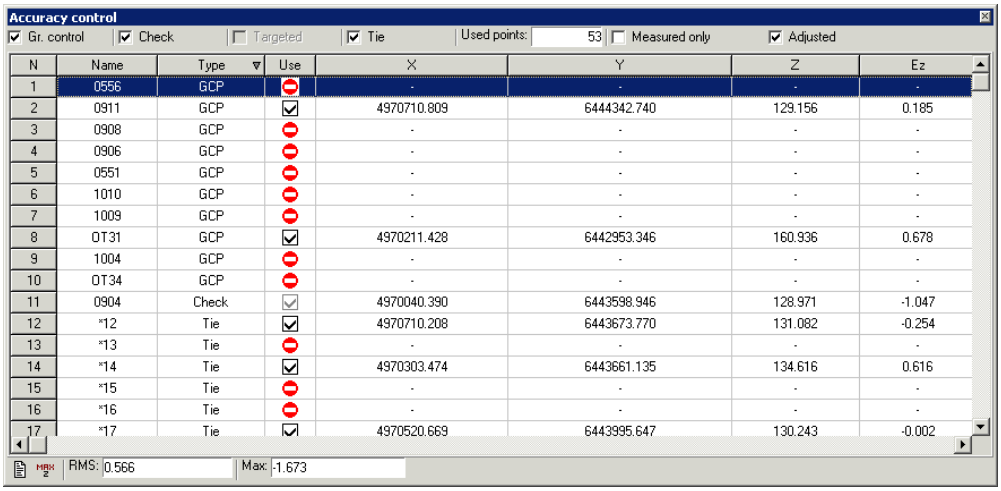
7.8.6. DEM control by triangulation points

The system provides possibility to control DEM creation quality by comparing DEM vertices coordinates with coordinates of points located in triangulation points layer. This triangulation points layer should not be used during this DEM creation.

To do this use the **TIN › Check accuracy › Check against triangulation points** menu item.

It is not necessary to create points vector layer.

The **Accuracy control** window includes a table of all points (GCP/check/tie), acquired on on aerial triangulation step (see the 'Aerial triangulation' User Manual).



N	Name	Type	Use	X	Y	Z	Ez
1	0556	GCP	⛔	-	-	-	-
2	0911	GCP	☑	4970710.809	6444342.740	129.156	0.185
3	0908	GCP	⛔	-	-	-	-
4	0906	GCP	⛔	-	-	-	-
5	0551	GCP	⛔	-	-	-	-
6	1010	GCP	⛔	-	-	-	-
7	1009	GCP	⛔	-	-	-	-
8	OT31	GCP	☑	4970211.428	6442953.346	160.936	0.678
9	1004	GCP	⛔	-	-	-	-
10	OT34	GCP	⛔	-	-	-	-
11	0904	Check	☑	4970040.390	6443598.946	128.971	-1.047
12	*12	Tie	☑	4970710.208	6443673.770	131.082	-0.254
13	*13	Tie	⛔	-	-	-	-
14	*14	Tie	☑	4970303.474	6443661.135	134.616	0.616
15	*15	Tie	⛔	-	-	-	-
16	*16	Tie	⛔	-	-	-	-
17	*17	Tie	☑	4970520.669	6443995.647	130.243	-0.002

Fig. 118. Accuracy control using triangulation points

The top toolbar of the window is used to setup of triangulation points display in the list: **Gr.control**, **Check**, **Tie**, **Targeted**, **Measured only** (triangulation points with Z coordinate calculated during DEM creation), **Adjusted**.

The **Used points** field is used to display number of points used for calculation of DEM deviation by Z-coordinate (in the **Use** column the ☑ is set).

The table of triangulation points contains the following columns:

- `Name` – point number;
- `Type` – point type: GCP, check or tie point;
- `Use` – data about point use for calculating of DEM deviation by Z-coordinate:
  - ☑ – the triangulation point was used;
  - ☐ – the triangulation point was not used;
  - ⛔ – the triangulation point cannot be used, since it was not used for adjustment.
- `X, Y, Z` – adjusted coordinates of triangulation points;
- `Ez` – value of DEM deviation from the triangulation point by Z-coordinate.

Status panel is used to display values of root mean square error (`RMS`) and a value of maximal error of DEM deviation by Z-coordinate (`Max`).

The button allows to display in the list a point with maximal error of deviation from triangulation points by Z-coordinate.

The button allows to display detailed report on accuracy control.

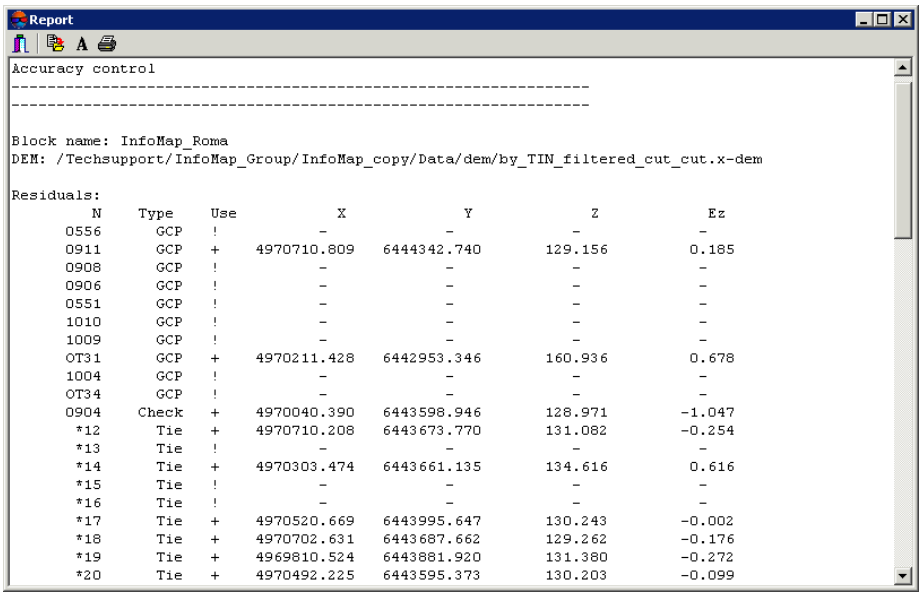


Fig. 119. Report on accuracy control using triangulation points

The top panel of the **Report** window contains the following buttons:

- – allows to close report window;
- – allows to save a report to file system of *Windows*;
- – allows to change font, style, size, and color of report text;
- – allows to print a report.

7.9. DEM saving

The system allows to save a DEM to file with `x-dem` extension. File name and target folder to place DEM in active profile resources should be specified prior to the DEM creation.

To save opened DEM to a new file the system provides the **DEM › Save copy...** menu item.

To save DEM area select a part of the DEM (see the chapter '*Vector objects selection*' in the 'Vectorization' User Manual) and choose **DEM › Save selection**.

To save a DEM as a raster file with geodetic reference perform the following actions:

1. Load DEM.

2. Select **DEM › Save as georeferenced raster...**. The **Settings** window opens.

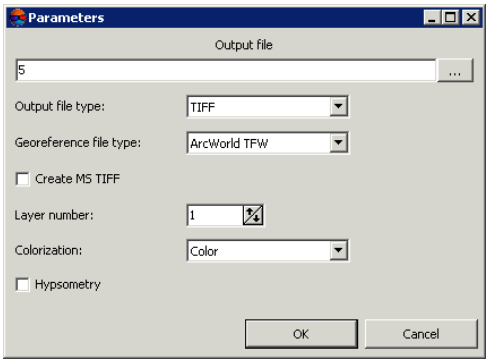


Fig. 120. Parameters of saving DEM to raster file

3. In the **Output file** section click the ... button and specify name and target folder of output DEM out active profile resources.

4. Select **Output file type** – TIFF or Windows BMP.

5. Select **Georeferenced file type**:

   - **No** – if there is no georeferenced files;
   - **PHOTOMOD GEO**;
   - **ArcWorld TFW**;
   - **MapInfo TAB**.

6. [optional] To create additional interior system file in MS TIFF format, that allows to use output raster file in the system without any conversion, set the **Create MS TIFF** checkbox on.

7. [optional] To save as an image the level of the pyramid that is different from the original, specify the **Layer number**.

8. Select the **Colorization** of output file – **Color** or **Grayscale**.

9. [optional] To increase the detail of the display when save the DEM as a raster file, set the **Hypsometry** checkbox on.

10. Click OK. After that the system creates file with raster representation of the DEM, and georeferenced file as well.

7.10. DEM recovering

The system provides possibility to recover DEM's consistency, which may be disrupted by applying of DEM editing or filtering operations.

To recover a DEM perform the following actions:

1. Load a DEM in the project.

2. Select **DEM › Recover**. The **Rebuild DEM pyramid** window opens.

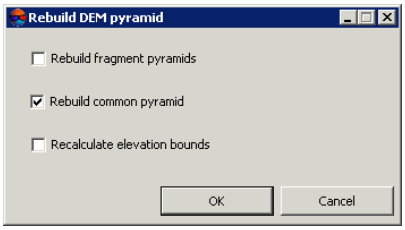


Fig. 121. DEM recovering

3. Select the following parameters of DEM recovering:

   - **Rebuild fragment pyramids** – allows to rebuild a DEM by fragments (tiles) if the DEM consist of them;
   - **Rebuild common pyramid** (is set on by default) – allows to rebuild common pyramid for DEM;
   - **Recalculate elevation bounds** – allows to change DEM colour representation after changing its boundaries elevation.

4. Click OK. After that the system rebuilds DEM and shows a message about successful or unsuccessful DEM update.

7.11. DEM loading

The system allows to load saved DEM both from active profile resources and from file that is out of active profile resources.

In order to load a DEM from active profile resources, do the following:

1. Select **DEM| Load** (**Ctrl+O, D**) or click the buttons of the main toolbar. The **Open** window opens.

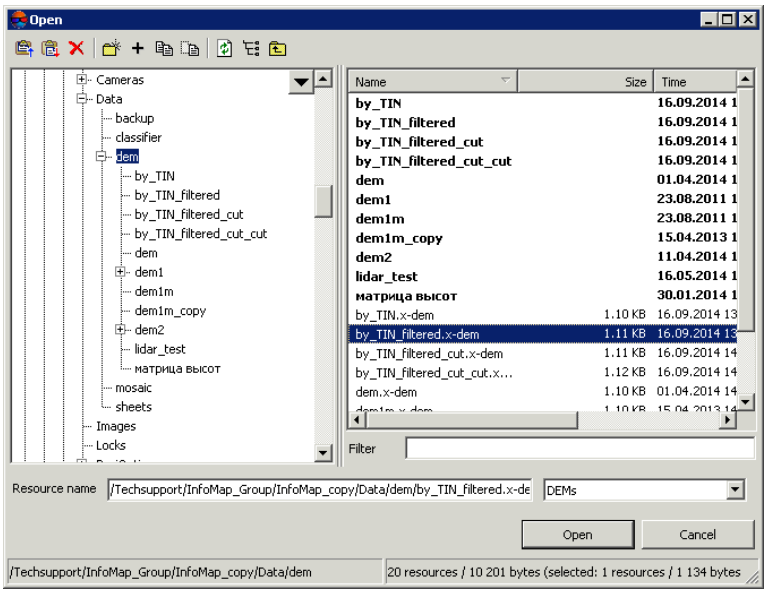


Fig. 122. DEM loading

2. Select one or multiple DEM files in active profile resources with `x-dem` extension.

When multiple files are selected, each DEM is loaded to separate layer.

3. Click **Open**. If one or multiple DEM containing layers are already loaded to active layer, then during loading of a new layer the window of loading type is opened.

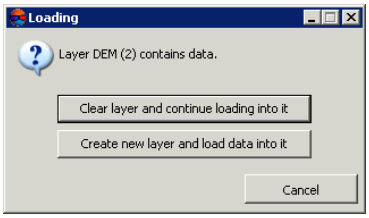


Fig. 123. DEM loading

The system provides the following possibilities of new layer loading to existing DEM layer:

- **Clear layer and continue loading into it** – data of active layer is replaced by data from loading layer;
- **Create new layer and load data into it** – data is loaded to a new layer.

Corresponding *DEM* layers are displayed in *Manager*.

To load DEM from external data without its conversion to internal format the **DEM › Open from file...** menu item is used.

DEM loading and opening in format with extension different from `x-dem` may take a lot of time.

The system also allows to transfer lidar data to a DEM (see the 'LIDAR Data processing' User Manual).

7.12. DEM export

7.12.1. Convert DEM to pickets

The system allows to convert DEM cells pickets with specified simplifying.

To do this conversion perform the following actions:

1. Load or create a DEM.
2. [optional] Load vector layer and select polygons, within which it is necessary to perform the conversion.
3. Select **DEM › Convert to points...**. The **Parameters of converting to points** window opens.

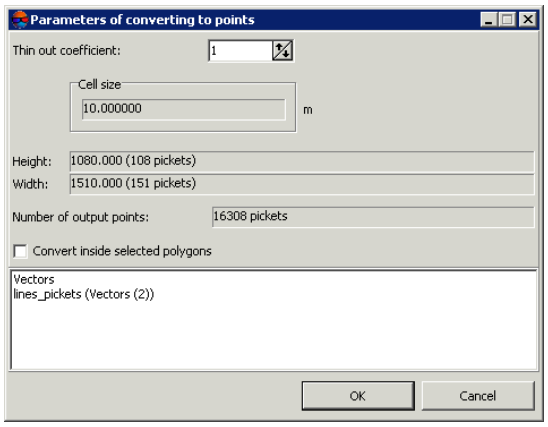


Fig. 124. Parameters of DEM converting to pickets

4. Specify **Thinout coefficient** of DEM.

   Depending on this coefficient the system calculates the **Cell size** of output DEM in meters, **Height** and **Width** in meters and in pickets number. Besides, the estimated **Number of output points** is displayed in this window.

5. [optional] If additional vector layer with polygons is loaded, set the **Convert inside selected polygons** checkbox on and select a layer with polygons in the list to perform conversion in specified areas only.

6. Click OK. When the conversion is completed the system displays a message about number of acquired pickets and new vector layer with obtained pickets is created.

7.12.2. DEM export

The system provides possibility of DEM export to the following formats.

- **ArcInfo ASCII grid** – exchange format with `asc` or `grd` extension, that is used in the program *ArcInfo;*
- **ASCII** – exchange text format with `txt` extension, that is supported by a large number of programs with different specializations (see description of the format in the 'Vectorization') User Manual;
- **Panorama MTW** – exchange format with `mtw`extension, that is used in the program *GIS Map 2000*;
- **SurferASCII grid** – exchange format with `grd`extension, that is used in the program *Surfer*;
- **USGS DEM** – exchange format with `dem` extension, developed in United States Geological Survey (USGS).

To export DEM, for example, to ArcInfo ASCII grid format choose **DEM › Export › ArcInfo ASCII grid...**. The **Export to ArcInfo grid** window opens. Choose a folder to place a file in *Windows* file system. Specify a file name in the **File name** input field. Click the **Save** button to complete the export.

7.12.3. Export to CSV

The system provides possibility of DEM export to CSV format. CSV format is exchange text format with `csv` extension, which is supported by major applications in different industries. It is used as exchange format when special geospatial data formats are not applicable for some reason.

In order to export DEM to CSV format perform the following actions:

1. Select **DEM › Export › CSV**. The **Export to CSV** window opens.

2. Choose a folder to place a file to file system of *Windows*.

3. Click the **Save** button. The **Export to CSV** window opens.

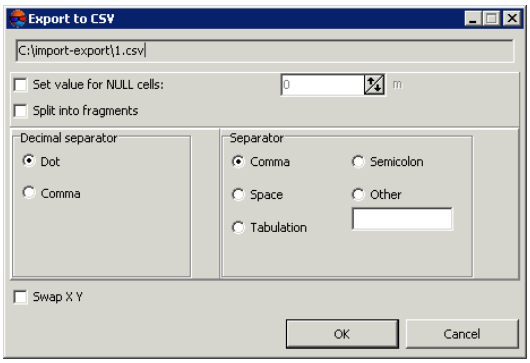


Fig. 125. Parameters of export to CSV format

4. [optional] To fill DEM null cells set the **Set value for null cells** checkbox on and input necessary value to the input field in meters.

5. [optional] To split CSV-file into separate parts, set the **Split into fragments** checkbox on.

6. In the **Decimal separator** section select point or comma to be used to separate coordinates.

7. In the **Field separator** section choose, what is used to separate fields: comma, space, tab, semicolon or other delimiters.

   If the **Field separator** is selected as a comma, it is not recommended to specify the **Decimal separator** as a comma, since objects with incorrect coordinates will be created after export operation.

8. [optional] In order to swap coordinate system (left/right) during DEM export, set the **Swap X Y** checkbox.

   If the **Swap X Y** checkbox is off, the initial data will be imported in the right coordinate system. Otherwise, in the left one.

9. Click OK to complete the export operation.

7.12.4. Export to DTED

The system provides possibility of DEM export to Digital Terrain Elevation Data (DTED) format. This is exchange format with `dt2` extension, that is used in the following programs: *ESRI ArcGIS Desktop* and *Pitney Bowes MapInfo*.

In order to export DEM to DTED format perform the following actions:

1. Select **DEM › Export › DTED**. The **Export to DTED** window opens.

2. Choose a folder to place a file in *Windows* file system.

3. Specify a file name in the **File name** input field.

4. Click the **Save** button. The **Settings** window opens.

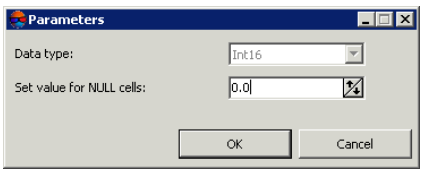


Fig. 126. Parameters of export to DTED format

To export the data type having an integer value in the **Data type** field the **Int16** parameters is set by default (integer data type is used to represent numbers in the range from -32 768 to 32 767).

5. [optional] To fill DEM null cells specify the **Set value for null cells** parameter in meters.

6. Click OK to complete the export operation.

7.12.5. Export to ERDAS Imagine

The system provides possibility of DEM export to ERDAS Imagine format. This is exchange format with `img` extension, that is used in the following programs: *ERDAS Imagine* and *ESRI ArcGIS Desktop*.

In order to export DEM to ERDAS Imagine format perform the following actions:

1. Select **DEM › Export › ERDAS Imagine**. The **Export to ERDAS Imagine** window opens.

2. Choose a folder to place a file in *Windows* file system.

3. Specify a file name in the **File name** input field.

4. Click the **Save** button. The **Settings** window opens.

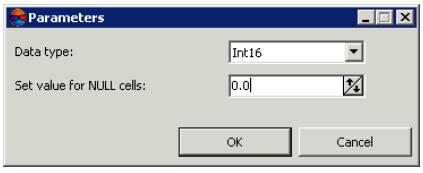


Fig. 127. Parameters of export to ERDAS Imagine format

5. Choose in the **Data type** list one of the following data types:

   - **UInt 8** – integer data type, that is used to represent numbers in the range from 0 to 255;
   - **Int 8** – integer data type, that is used to represent numbers in the range from -128 to 127;
   - **UInt 16** – integer data type, that is used to represent numbers in the range from 0 to 65535;
   - **Int 16** – integer data type, that is used to represent numbers in the range from -32,768 to 32,767;
   - **UInt 32** – integer data type, that is used to represent numbers in the range from 0 to 4,294,967,295;
   - **Int 32** – integer data type, that is used to represent numbers in the range from -2,147,483,648 to 2,147,483,647;
   - **Float 32** – data type, that is used to represent numbers with floating point in the range from 3.4e-038 to 3.4e+038;
   - **Float 64** – data type, that is used to represent numbers with floating point in the range from 1.7e-308 to 1.7e+308.

6. [optional] To fill DEM null cells specify the **Set value for null cells** parameter in meters.

7. Click OK to complete the export operation.

7.12.6. Export to GeoTIFF

The system allows to export DEM to GeoTIFF format, that is exchange format with `tif` extension. GeoTIFF format – is common format for storing georeferenced images.

Information about used coordinates system for DEM are displayed as a code of European Petroleum Survey Group (EPSG) standard.

In order to export DEM to GeoTIFF format perform the following actions:

1. Select **DEM › Export › GeoTIFF**. The **Export to GeoTIFF** window opens.

2. Choose a folder to place a file in *Windows* file system.

3. Specify a file name in the **File name** input field.

4. Click the **Save** button. The **Settings** window opens.

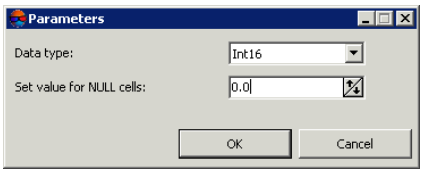


Fig. 128. Parameters of export to GeoTIFF format

5. Choose in the **Data type** list one of the following data types:

   - **UInt 8** – integer data type, that is used to represent numbers in the range from 0 to 255;
   - **Int 8** – integer data type, that is used to represent numbers in the range from -128 to 127;
   - **UInt 16** – integer data type, that is used to represent numbers in the range from 0 to 65535;
   - **Int 16** – integer data type, that is used to represent numbers in the range from -32,768 to 32,767;
   - **UInt 32** – integer data type, that is used to represent numbers in the range from 0 to 4,294,967,295;
   - **Int 32** – integer data type, that is used to represent numbers in the range from -2,147,483,648 to 2,147,483,647;
   - **Float 32** – data type, that is used to represent numbers with floating point in the range from 3.4e-038 to 3.4e+038;
   - **Float 64** – data type, that is used to represent numbers with floating point in the range from 1.7e-308 to 1.7e+308.

6. [optional] To fill DEM null cells specify the **Set value for null cells** parameter in meters.

7. Click OK to complete the export operation.

7.12.7. Export to PCIDSK

The system allows to export DEM to PCIDSK format, that is exchange format with `pix` extension and is used in the following program: *PCI Geomatics Geomatica*.

In order to export DEM to PCIDSK format perform the following actions:

1. Select **DEM › Export › PCIDSK**. The **Export to PCIDSK** window opens.

2. Choose a folder to place a file in *Windows*.

3. Specify a file name in the **File name** input field.

4. Click the **Save** button. The **Settings** window opens.

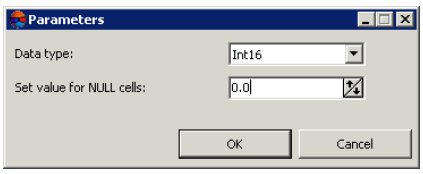


Fig. 129. Parameters of export to PCIDSK format

5. Choose in the **Data type** list one of the following data types:

   - **UInt 8** – integer data type, that is used to represent numbers in the range from 0 to 255;
   - **Int 8** – integer data type, that is used to represent numbers in the range from -128 to 127;
   - **UInt 16** – integer data type, that is used to represent numbers in the range from 0 to 65535;
   - **Int 16** – integer data type, that is used to represent numbers in the range from -32,768 to 32,767;
   - **UInt 32** – integer data type, that is used to represent numbers in the range from 0 to 4,294,967,295;
   - **Int 32** – integer data type, that is used to represent numbers in the range from -2,147,483,648 to 2,147,483,647;
   - **Float 32** – data type, that is used to represent numbers with floating point in the range from 3.4e-038 to 3.4e+038;
   - **Float 64** – data type, that is used to represent numbers with floating point in the range from 1.7e-308 to 1.7e+308.

6. [optional] To fill DEM null cells specify the **Set value for null cells** parameter in meters.

7. Click OK to complete the export operation.

7.13. DEM import

The system allows to import DEM from files with the following extensions: `grd`, `asc`, `tif`, `dem`, `mtw`, `dt1`, `dt2`, `img`, `pix`, `hgt`.

In order to perform import of DEM from a file of supported format, do the following:

1. Select **DEM › Import**. The **DEM import** window opens.
2. In the file system of *Windows* select a DEM file.
3. Click the **Open** button. The **DEM import** window opens.

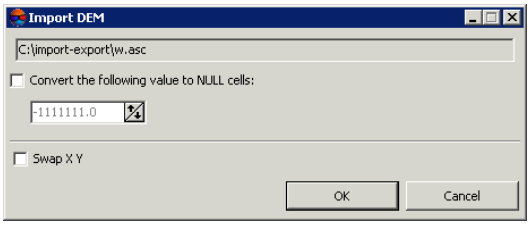


Fig. 130. Parameters of DEM import

4. [optional] To convert DEM cells with specified values to null cells, set the **Convert the following value to null cells** checkbox and define necessary value.
5. [optional] To prevent creating text attributes with an empty value during import, set the **Do not create empty text attributes** checkbox on.

   When the **Do not create empty text attributes** checkbox is on, in some cases a data loss may occur.

6. [optional] In order to swap coordinate system during DEM import, set the **Swap X Y** checkbox on.

   If the **Swap X Y** checkbox is set off, the initial data will be imported in the right coordinate system. Otherwise, in the left one.

7. Click OK. The **Destination file** window opens.

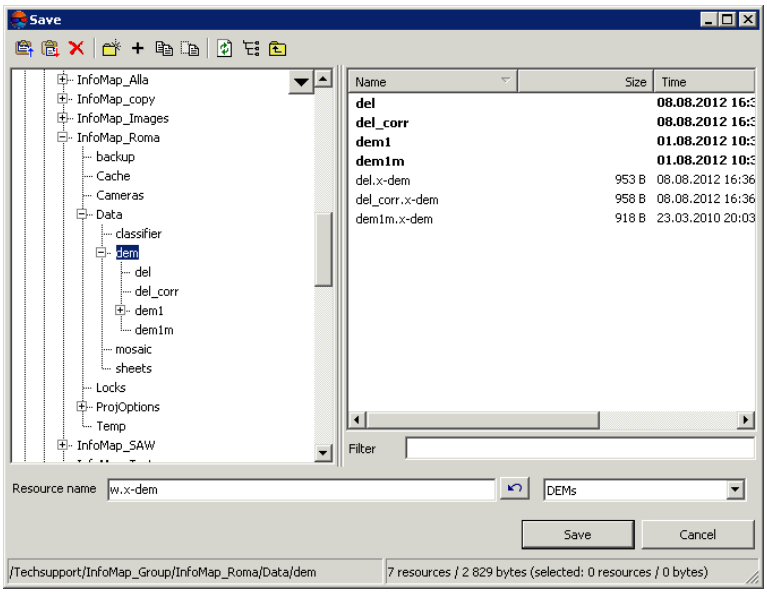


Fig. 131. Imported DEM saving

8. Select target file placement in active profile resources and click **Save**.

The system allows to import DEM in batch mode from files with the following extensions: `grd`, `asc`, `tif`, `dem`, `mtw`, `dt1`, `dt2`, `img`, `pix`, `hgt`. During batch import the system loads multiple files from specified folder at once.

In order to import necessary number of layers, containing DEMs, from specified folder, perform the following actions:

1. Select **DEM › Batch import**. The **DEM import** window opens.
2. In the file system of *Windows* select multiple files with DEM.
3. Click the **Open** button. The **DEM import** window opens.

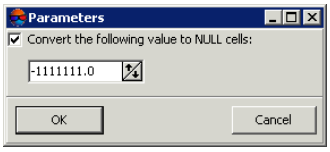


Fig. 132. Parameters of DEM import

4. [optional] To convert DEM cells with specified values to null cells, set the **Convert the following value to null cells** checkbox and define necessary value.
5. Click OK. The **Select folder** window opens.

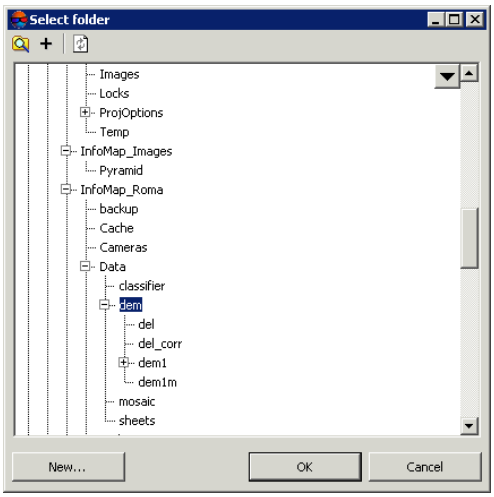


Fig. 133. Imported DEM saving

6. Select a folder or create a new one using the **Create...** button.

7. Click OK to complete the import operation.